Proprietary notice

Copyright (c) 2014 Schlumberger. All rights reserved. Reproduction or alteration without prior written permission is prohibited, except as allowed under applicable law.

Use of this product is governed by the License Agreement. Schlumberger makes no warranties, express, implied, or statutory, with respect to the product described herein and disclaims without limitations any warranties of merchantability or fitness for a particular purpose.

Trademarks & service marks

"Schlumberger," the Schlumberger logotype, and other words or symbols used to identify the products and services described herein are either trademarks, trade names, or service marks of Schlumberger and its licensors, or are the property of their respective owners. These marks may not be copied, imitated, or used, in whole or in part, without the express prior written permission of their owners. In addition, covers, page headers, custom graphics, icons, and other design elements may be service marks, trademarks, and/or trade dress of Schlumberger and may not be copied, imitated, or used, in whole or in part, without the express prior written permission of Schlumberger.
Table of Contents

1 New Developments ...................................................................................... 1
   Licensing changes ............................................................................................. 1
   New facilities ........................................................................................................ 2
      ECLIPSE 100 .................................................................................................. 2
      ECLIPSE 300 ................................................................................................. 4
   Behavioral changes ............................................................................................ 7
      ECLIPSE 100 .................................................................................................. 7
      ECLIPSE 300 ................................................................................................. 10
   New keywords .................................................................................................... 13
      ECLIPSE 100 .................................................................................................. 13
      ECLIPSE 300 ................................................................................................. 15
   Altered keywords ............................................................................................... 16
      ECLIPSE 100 .................................................................................................. 16
      ECLIPSE 300 ................................................................................................. 20

2 Data File Overview ..................................................................................... 24
   Data file structure .............................................................................................. 26
      Keyword data .................................................................................................. 26
      Comments ........................................................................................................ 27
   Global keywords ................................................................................................. 27
   ECLIPSE 300 error handling .............................................................................. 29
   Example data file ............................................................................................... 30
   RUNSPEC section ............................................................................................. 31
      Alphabetic list of RUNSPEC keywords .......................................................... 32
   GRID section ..................................................................................................... 38
      Rock permeability values .................................................................................. 41
      Transmissibility multipliers .............................................................................. 42
      Diffusivity multipliers ....................................................................................... 42
      BOX and ENDBOX keywords ........................................................................ 42
      Non-neighbor connections (NNCs) .................................................................. 43
      Output control .................................................................................................. 44
      Local grid refinement ....................................................................................... 45
      Description of GRID section keywords .......................................................... 45
      Transmissibility keywords .............................................................................. 47
      Transmissibility modifiers ............................................................................... 48
      Faults ............................................................................................................... 48
      Numerical aquifers ........................................................................................... 49
      Dual porosity .................................................................................................... 49
Compressed vertical equilibrium (ECLIPSE 100) .......................................................... 50
Diffusivity modifiers ........................................................................................................ 50
Independent reservoir regions (ECLIPSE 100) ............................................................. 51
Temperature option (ECLIPSE 100) .............................................................................. 51
Nine-point scheme (ECLIPSE 300) ............................................................................... 52
Operational keywords .................................................................................................... 52
Local grid refinement ..................................................................................................... 53
Tensor permeability ....................................................................................................... 54
Geomechanics (ECLIPSE 300) ..................................................................................... 55
Alphabetical list of GRID section keywords ................................................................... 55
Example data set ........................................................................................................... 62

EDIT section .................................................................................................................... 63
Changing cell depths ..................................................................................................... 64
Changing pore volumes ................................................................................................. 64
Changing transmissibilities ............................................................................................ 64
Changing diffusivities ..................................................................................................... 64
Modifying non-neighbor connections ............................................................................. 64
Output control ................................................................................................................ 65
Alphabetical list of EDIT section keywords ................................................................. 65

PROPS section ................................................................................................................ 68
Saturation functions ....................................................................................................... 69
  Two dimensional models for saturation functions (for three phase simulations) ..... 70
  Stability issues ........................................................................................................... 71
Hysteresis ...................................................................................................................... 72
End point scaling .......................................................................................................... 72
Pressure tables .............................................................................................................. 72
Rock pressure tables ..................................................................................................... 73
Water pressure tables .................................................................................................... 73
Surface densities ........................................................................................................... 74
Oil and gas pressure tables (black oil) .......................................................................... 74
Essential keywords - black oil case ............................................................................... 74
Standard conditions (ECLIPSE 300) ............................................................................. 76
Compositional Equation of State data (ECLIPSE 300) .................................................. 76
K-value tables (ECLIPSE 300) ...................................................................................... 77
Diffusion coefficients .................................................................................................... 79
Composition with respect to pressure tables (ECLIPSE 300) ....................................... 79
PROPS section keywords ............................................................................................. 79
  Alkaline option .......................................................................................................... 79
  API tracking (ECLIPSE 100) ...................................................................................... 80
  Black Oil fluid PVT property data ............................................................................ 80
  Brine option .............................................................................................................. 81
  Carbon Dioxide solution in aqueous phase (ECLIPSE 300) ..................................... 81
  Carter-Tracy aquifer data .......................................................................................... 82
  Chemical reaction keywords (ECLIPSE 300) .......................................................... 82
  Coal bed methane (ECLIPSE 100) ............................................................................ 82
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Example SUMMARY section</td>
<td>152</td>
</tr>
<tr>
<td>Summary file names</td>
<td>153</td>
</tr>
<tr>
<td>Well and group flow quantities</td>
<td>153</td>
</tr>
<tr>
<td>Miscellaneous well and group quantities</td>
<td>170</td>
</tr>
<tr>
<td>Secondary group and well quantities (ECLIPSE 300)</td>
<td>176</td>
</tr>
<tr>
<td>Well list quantities</td>
<td>177</td>
</tr>
<tr>
<td>Grid block and region quantities</td>
<td>178</td>
</tr>
<tr>
<td>Equivalent directional keywords for relative permeabilities (ECLIPSE 100)</td>
<td>189</td>
</tr>
<tr>
<td>Processed scaled end-points</td>
<td>190</td>
</tr>
<tr>
<td>Reservoir volumes</td>
<td>191</td>
</tr>
<tr>
<td>Oil recovery efficiencies</td>
<td>192</td>
</tr>
<tr>
<td>Oil recovery mechanism</td>
<td>192</td>
</tr>
<tr>
<td>Recovery plant options (ECLIPSE 300 only)</td>
<td>193</td>
</tr>
<tr>
<td>Analytic aquifer quantities</td>
<td>193</td>
</tr>
<tr>
<td>Numerical aquifer quantities</td>
<td>194</td>
</tr>
<tr>
<td>Tracer and API tracking quantities</td>
<td>194</td>
</tr>
<tr>
<td>Brine option quantities</td>
<td>195</td>
</tr>
<tr>
<td>Temperature option quantities</td>
<td>197</td>
</tr>
<tr>
<td>Environmental tracers</td>
<td>197</td>
</tr>
<tr>
<td>Coal Bed Methane Model (ECLIPSE 100)</td>
<td>199</td>
</tr>
<tr>
<td>Foam Model</td>
<td>199</td>
</tr>
<tr>
<td>Gas Field Operations Model</td>
<td>200</td>
</tr>
<tr>
<td>Simulator performance keyword</td>
<td>201</td>
</tr>
<tr>
<td>Gas lift optimization facility</td>
<td>201</td>
</tr>
<tr>
<td>Gas calorific value control option</td>
<td>202</td>
</tr>
<tr>
<td>Gi pseudo-compositional model</td>
<td>202</td>
</tr>
<tr>
<td>Local grid refinement and coarsening</td>
<td>203</td>
</tr>
<tr>
<td>Operational keywords</td>
<td>203</td>
</tr>
<tr>
<td>Plotting local timestep results in GRAF</td>
<td>203</td>
</tr>
<tr>
<td>Multisegment wells</td>
<td>204</td>
</tr>
<tr>
<td>Rivers</td>
<td>206</td>
</tr>
<tr>
<td>Network option</td>
<td>208</td>
</tr>
<tr>
<td>Polymer Flood Model</td>
<td>209</td>
</tr>
<tr>
<td>Pseudo Steady-State (ECLIPSE 100 only)</td>
<td>212</td>
</tr>
<tr>
<td>Reservoir coupling facility (ECLIPSE 100 only)</td>
<td>213</td>
</tr>
<tr>
<td>Solvent Model</td>
<td>213</td>
</tr>
<tr>
<td>Surfactant Model</td>
<td>213</td>
</tr>
<tr>
<td>Alkaline model</td>
<td>214</td>
</tr>
<tr>
<td>Wellbore friction model</td>
<td>215</td>
</tr>
<tr>
<td>Conductive Faults</td>
<td>215</td>
</tr>
<tr>
<td>User defined quantities</td>
<td>216</td>
</tr>
<tr>
<td>User defined quantities</td>
<td>216</td>
</tr>
<tr>
<td>ODD3P three-phase relative permeability and capillary pressure hysteresis model</td>
<td>216</td>
</tr>
<tr>
<td>Asphaltene model</td>
<td>219</td>
</tr>
<tr>
<td>Simulator performance keywords</td>
<td>219</td>
</tr>
<tr>
<td>Other keywords</td>
<td>223</td>
</tr>
</tbody>
</table>
Deprecated keywords .................................................................................................. 224
Deleted keywords ........................................................................................................ 225

SCHEDULE section ....................................................................................................... 226
Define wells, groups and well connections .............................................................. 227
  Define well connections and set connection properties ............................................. 228
  Define multisegment wells ....................................................................................... 229
  Define separators (ECLIPSE 300) ........................................................................... 231
Well controls and limits ................................................................................................. 231
  Production well control data ..................................................................................... 232
  Injection well control data .......................................................................................... 232
  Vertical flow performance keywords ........................................................................ 234
  Well economic limits and coning control ................................................................. 235
Group controls and limits ............................................................................................. 235
  Group production control by guide rate ................................................................. 235
  Group production control by prioritization .............................................................. 236
  Group economic limits and coning control ............................................................... 236
  Group injection control ............................................................................................. 236
  Group efficiency factor ............................................................................................. 237
  Sales gas control (ECLIPSE 100) ............................................................................ 237
  Satellite group flows ................................................................................................. 238
  Sustainable capacity test .......................................................................................... 238
Automatic drilling and workover controls ................................................................. 238
Action keyword facility ................................................................................................. 239
Periodic testing and cycling wells ............................................................................... 239
  Artificial lift operations (ECLIPSE 100) .................................................................... 239
Scale deposition model ................................................................................................. 240
Flash transformation and separator conditions ......................................................... 240
Gas processing (ECLIPSE 300) ................................................................................... 241
Aquifer controls ............................................................................................................ 241
Dissolved gas and vaporized oil constraints (ECLIPSE 100) ...................................... 242
Geomechanics option (ECLIPSE 300) ........................................................................ 242
Grid block property modifications .............................................................................. 242
Dynamic Regions Facility ............................................................................................ 243
Dual porosity .................................................................................................................. 243
Lagrangian Tracer Solver (ECLIPSE 300) ................................................................. 244
Simulator controls ........................................................................................................ 244
  User defined values .................................................................................................. 245
Output .......................................................................................................................... 245
Conductive Fractures (ECLIPSE 300) ......................................................................... 246
Adjoint gradient screening option (ECLIPSE 300) .................................................... 246
Advance simulation to new time/date ......................................................................... 246
Instantaneous gradient option ..................................................................................... 247
Polymer flood model .................................................................................................... 247

OPTIMIZE section ......................................................................................................... 247
OPTIMIZE section keywords ...................................................................................... 248
3 Keywords ................................................................................................................. 249

A ...................................................................................................................................... 249

ACF .............................................................................................................................. 250
    Examples ......................................................................................................................... 250
ACFDET ......................................................................................................................... 251
    Examples ......................................................................................................................... 251
ACFS .............................................................................................................................. 252
ACTCO2S ....................................................................................................................... 253
ACTDIMS ....................................................................................................................... 254
ACTION .......................................................................................................................... 255
ACTIONG ....................................................................................................................... 258
ACTIONR ....................................................................................................................... 263
ACTIONS ......................................................................................................................... 266
ACTIONW ....................................................................................................................... 270
ACTIONX ......................................................................................................................... 275
    Quantity types and definitions ..................................................................................... 277
    Notes ............................................................................................................................. 281
    Examples ......................................................................................................................... 282
ACTNUM ......................................................................................................................... 285
ACTPARAM ..................................................................................................................... 287
ADD ............................................................................................................................... 288
    Legal arrays .................................................................................................................. 289
    Notes ............................................................................................................................. 290
    Examples ......................................................................................................................... 291
ADDREG ......................................................................................................................... 292
    Legal arrays .................................................................................................................. 293
    Notes ............................................................................................................................. 294
    Additional notes ............................................................................................................. 295
    Examples ......................................................................................................................... 295
ADDZCORN .................................................................................................................... 297
    Examples ......................................................................................................................... 297
ADSALNOD ................................................................................................................... 300
ADSORP .......................................................................................................................... 301
AIM ................................................................................................................................. 304
AIMCON .......................................................................................................................... 305
AIMFRAC ........................................................................................................................ 308
AIMPVI ............................................................................................................................ 309
AJGPARAM ..................................................................................................................... 310
AJGRADNT ..................................................................................................................... 311
AJGWELLS ..................................................................................................................... 312
ALKADS ......................................................................................................................... 313
ALKALINE ...................................................................................................................... 314
ALKROCK ....................................................................................................................... 315
ALPHA ......................................................................................................................... 316
ALPHAD .......................................................................................................................... 317
<table>
<thead>
<tr>
<th>Page</th>
<th>Content</th>
</tr>
</thead>
<tbody>
<tr>
<td>383</td>
<td>Examples</td>
</tr>
<tr>
<td>384</td>
<td>Examples</td>
</tr>
<tr>
<td>385</td>
<td>Example</td>
</tr>
<tr>
<td>386</td>
<td>Examples</td>
</tr>
<tr>
<td>388</td>
<td>Examples</td>
</tr>
<tr>
<td>389</td>
<td></td>
</tr>
<tr>
<td>390</td>
<td>Examples</td>
</tr>
<tr>
<td>394</td>
<td></td>
</tr>
<tr>
<td>395</td>
<td></td>
</tr>
<tr>
<td>396</td>
<td>BAKER1</td>
</tr>
<tr>
<td>397</td>
<td>Examples</td>
</tr>
<tr>
<td>399</td>
<td>Examples</td>
</tr>
<tr>
<td>401</td>
<td></td>
</tr>
<tr>
<td>402</td>
<td></td>
</tr>
<tr>
<td>403</td>
<td></td>
</tr>
<tr>
<td>404</td>
<td>BICAQ1</td>
</tr>
<tr>
<td>405</td>
<td>BICAQ2</td>
</tr>
<tr>
<td>406</td>
<td>BICAQ3</td>
</tr>
<tr>
<td>407</td>
<td>BICS</td>
</tr>
<tr>
<td>408</td>
<td>BIGMODEL</td>
</tr>
<tr>
<td>409</td>
<td>BIOTC</td>
</tr>
<tr>
<td>410</td>
<td>BIOTCTAB</td>
</tr>
<tr>
<td>412</td>
<td>BLACKOIL</td>
</tr>
<tr>
<td>413</td>
<td>BOGI</td>
</tr>
<tr>
<td>414</td>
<td>BOUNDARY</td>
</tr>
<tr>
<td>416</td>
<td>BOX</td>
</tr>
<tr>
<td>417</td>
<td>BPARA</td>
</tr>
<tr>
<td>418</td>
<td>BPIDIMS</td>
</tr>
<tr>
<td>419</td>
<td>BRANPROP</td>
</tr>
<tr>
<td>421</td>
<td>BRINE</td>
</tr>
<tr>
<td>422</td>
<td>BTOTALBA</td>
</tr>
<tr>
<td>423</td>
<td>BTOTALBFV</td>
</tr>
<tr>
<td>424</td>
<td></td>
</tr>
<tr>
<td>425</td>
<td>CALTRAC</td>
</tr>
<tr>
<td>426</td>
<td>CALVAL</td>
</tr>
<tr>
<td>427</td>
<td>CALVALR</td>
</tr>
<tr>
<td>Command</td>
<td>Page</td>
</tr>
<tr>
<td>---------</td>
<td>------</td>
</tr>
<tr>
<td>CARFIN</td>
<td>428</td>
</tr>
<tr>
<td>CART</td>
<td>430</td>
</tr>
<tr>
<td>CATYPE</td>
<td>431</td>
</tr>
<tr>
<td>CBMOPTS</td>
<td>432</td>
</tr>
<tr>
<td>CCTYPE</td>
<td>434</td>
</tr>
<tr>
<td>CCTYPES</td>
<td>435</td>
</tr>
<tr>
<td>CECON</td>
<td>436</td>
</tr>
<tr>
<td>CECONT</td>
<td>439</td>
</tr>
<tr>
<td>CECONT</td>
<td>441</td>
</tr>
<tr>
<td>CFLLIMIT</td>
<td>442</td>
</tr>
<tr>
<td>CGDTYPE</td>
<td>444</td>
</tr>
<tr>
<td>CGVTYPE</td>
<td>445</td>
</tr>
<tr>
<td>CHANDIMS</td>
<td>446</td>
</tr>
<tr>
<td>CIRCLE</td>
<td>448</td>
</tr>
<tr>
<td>CNAMES</td>
<td>449</td>
</tr>
<tr>
<td>CO2SOL</td>
<td>450</td>
</tr>
<tr>
<td>CO2STORE</td>
<td>451</td>
</tr>
<tr>
<td>COAL</td>
<td>452</td>
</tr>
<tr>
<td>COALADS</td>
<td>453</td>
</tr>
<tr>
<td>COALNUM</td>
<td>455</td>
</tr>
<tr>
<td>COALNUMR</td>
<td>456</td>
</tr>
<tr>
<td>COALAPP</td>
<td>458</td>
</tr>
<tr>
<td>COARSEN</td>
<td>460</td>
</tr>
<tr>
<td>CODTYPE</td>
<td>462</td>
</tr>
<tr>
<td>COLLAPSE</td>
<td>463</td>
</tr>
<tr>
<td>COLUMNS</td>
<td>464</td>
</tr>
<tr>
<td>COMPAGH</td>
<td>465</td>
</tr>
<tr>
<td>COMPAGHL</td>
<td>467</td>
</tr>
<tr>
<td>COMPDAT</td>
<td>469</td>
</tr>
<tr>
<td>COMPDATL</td>
<td>474</td>
</tr>
<tr>
<td>COMPDATM</td>
<td>479</td>
</tr>
<tr>
<td>COMPFLASH</td>
<td>480</td>
</tr>
<tr>
<td>COMPIMB</td>
<td>484</td>
</tr>
<tr>
<td>COMPINJK</td>
<td>486</td>
</tr>
<tr>
<td>COMPKRI</td>
<td>488</td>
</tr>
<tr>
<td>COMPKRIL</td>
<td>490</td>
</tr>
<tr>
<td>COMPLMPL</td>
<td>492</td>
</tr>
<tr>
<td>COMPLUMP</td>
<td>495</td>
</tr>
<tr>
<td>COMPMBIL</td>
<td>497</td>
</tr>
<tr>
<td>COMPROMOBI</td>
<td>499</td>
</tr>
</tbody>
</table>

**Treatment of flash transformation ratios in ECLIPSE**
<table>
<thead>
<tr>
<th>Command</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>DIFFMZ</td>
<td>624</td>
</tr>
<tr>
<td>DIFFMZ-</td>
<td>625</td>
</tr>
<tr>
<td>DIFFR</td>
<td>626</td>
</tr>
<tr>
<td>DIFFFGAS</td>
<td>627</td>
</tr>
<tr>
<td>DIFFTHT</td>
<td>628</td>
</tr>
<tr>
<td>DIFFTOIL</td>
<td>629</td>
</tr>
<tr>
<td>DIFFUSE</td>
<td>630</td>
</tr>
<tr>
<td>DIFFX</td>
<td>631</td>
</tr>
<tr>
<td>DIFFY</td>
<td>632</td>
</tr>
<tr>
<td>DIFFZ</td>
<td>633</td>
</tr>
<tr>
<td>DIMENS</td>
<td>634</td>
</tr>
<tr>
<td>DIMPES</td>
<td>635</td>
</tr>
<tr>
<td>DIMPLICT</td>
<td>636</td>
</tr>
<tr>
<td>DISGAS</td>
<td>637</td>
</tr>
<tr>
<td>DISPBC</td>
<td>638</td>
</tr>
<tr>
<td>DISPBCL</td>
<td>640</td>
</tr>
<tr>
<td>DSDIMS</td>
<td>642</td>
</tr>
<tr>
<td>DISPERSE</td>
<td>643</td>
</tr>
<tr>
<td>DNGL</td>
<td>645</td>
</tr>
<tr>
<td>DOMAIN</td>
<td>646</td>
</tr>
<tr>
<td>DOMAINS</td>
<td>647</td>
</tr>
<tr>
<td>DPCDT</td>
<td>648</td>
</tr>
<tr>
<td>DGRID</td>
<td>649</td>
</tr>
<tr>
<td>DPKRMODE</td>
<td>650</td>
</tr>
<tr>
<td>DPNUM</td>
<td>652</td>
</tr>
<tr>
<td>DR</td>
<td>653</td>
</tr>
<tr>
<td>DRAINAGE</td>
<td>654</td>
</tr>
<tr>
<td>DREF</td>
<td>655</td>
</tr>
<tr>
<td>DREFS</td>
<td>656</td>
</tr>
<tr>
<td>DREFT</td>
<td>657</td>
</tr>
<tr>
<td>DREFTS</td>
<td>658</td>
</tr>
<tr>
<td>DREFW</td>
<td>659</td>
</tr>
<tr>
<td>DREFW</td>
<td>660</td>
</tr>
<tr>
<td>DRLPRI</td>
<td>661</td>
</tr>
<tr>
<td>DRSDEI</td>
<td>662</td>
</tr>
<tr>
<td>DRSCT</td>
<td>664</td>
</tr>
<tr>
<td>DRV</td>
<td>665</td>
</tr>
<tr>
<td>DRVDT</td>
<td>666</td>
</tr>
<tr>
<td>DSPDEINT</td>
<td>667</td>
</tr>
<tr>
<td>DSTNUM</td>
<td>668</td>
</tr>
<tr>
<td>DTHETA</td>
<td>669</td>
</tr>
<tr>
<td>DTHETAV</td>
<td>670</td>
</tr>
<tr>
<td>DUALPERM</td>
<td>671</td>
</tr>
<tr>
<td>DUALPORO</td>
<td>672</td>
</tr>
<tr>
<td>Function</td>
<td>Page</td>
</tr>
<tr>
<td>--------------</td>
<td>------</td>
</tr>
<tr>
<td>ENPCVD</td>
<td>722</td>
</tr>
<tr>
<td>ENPCVT</td>
<td>723</td>
</tr>
<tr>
<td>ENPTVC</td>
<td>724</td>
</tr>
<tr>
<td>ENPTVD</td>
<td>726</td>
</tr>
<tr>
<td>ENPTVT</td>
<td>728</td>
</tr>
<tr>
<td>ENSPCVD</td>
<td>729</td>
</tr>
<tr>
<td>EOS</td>
<td></td>
</tr>
<tr>
<td>EOSNUM</td>
<td>730</td>
</tr>
<tr>
<td>EOSS</td>
<td>731</td>
</tr>
<tr>
<td>EPSCHECK</td>
<td>732</td>
</tr>
<tr>
<td>EPSCOMP</td>
<td>733</td>
</tr>
<tr>
<td>EPSDBGGS</td>
<td>734</td>
</tr>
<tr>
<td>EPSDEBUG</td>
<td>735</td>
</tr>
<tr>
<td>EPSODD3P</td>
<td>740</td>
</tr>
<tr>
<td>EQLDIMS</td>
<td>744</td>
</tr>
<tr>
<td>EQLDKVCR</td>
<td>750</td>
</tr>
<tr>
<td>EQLDREAC</td>
<td>751</td>
</tr>
<tr>
<td>EQLDTAB</td>
<td>753</td>
</tr>
<tr>
<td>EQLNUM</td>
<td>755</td>
</tr>
<tr>
<td>EQLOPTS</td>
<td>756</td>
</tr>
<tr>
<td>EQLZCORN</td>
<td>757</td>
</tr>
<tr>
<td>EQUALREG</td>
<td></td>
</tr>
<tr>
<td>Legal arrays</td>
<td>761</td>
</tr>
<tr>
<td>Notes</td>
<td>763</td>
</tr>
<tr>
<td>Examples</td>
<td>764</td>
</tr>
<tr>
<td>EQLS</td>
<td>765</td>
</tr>
<tr>
<td>Legal arrays</td>
<td>767</td>
</tr>
<tr>
<td>Notes</td>
<td>769</td>
</tr>
<tr>
<td>Examples</td>
<td>770</td>
</tr>
<tr>
<td>EQUIL</td>
<td>767</td>
</tr>
<tr>
<td>ESPLB</td>
<td>768</td>
</tr>
<tr>
<td>ESSNODE</td>
<td>774</td>
</tr>
<tr>
<td>EXCAVATE</td>
<td>775</td>
</tr>
<tr>
<td>EXCEL</td>
<td>776</td>
</tr>
<tr>
<td>EXFIN</td>
<td>777</td>
</tr>
<tr>
<td>EXTHOST</td>
<td>778</td>
</tr>
<tr>
<td>EXTRAPMS</td>
<td>779</td>
</tr>
<tr>
<td>EXTREPGL</td>
<td>780</td>
</tr>
<tr>
<td>F</td>
<td>781</td>
</tr>
<tr>
<td>FACTLI</td>
<td>782</td>
</tr>
<tr>
<td>Examples</td>
<td>783</td>
</tr>
<tr>
<td>FASTTRAC</td>
<td>784</td>
</tr>
<tr>
<td>FAULTDIM</td>
<td>785</td>
</tr>
<tr>
<td>Command</td>
<td>Page</td>
</tr>
<tr>
<td>--------------</td>
<td>------</td>
</tr>
<tr>
<td>GECON</td>
<td>920</td>
</tr>
<tr>
<td>GECON</td>
<td>923</td>
</tr>
<tr>
<td>GEFAC</td>
<td>926</td>
</tr>
<tr>
<td>GEODIMS</td>
<td>929</td>
</tr>
<tr>
<td>GEOMECH</td>
<td>930</td>
</tr>
<tr>
<td>GEOYCAP</td>
<td>932</td>
</tr>
<tr>
<td>GEOYLDI</td>
<td>933</td>
</tr>
<tr>
<td>GETDATA</td>
<td>935</td>
</tr>
<tr>
<td>GETGLOB</td>
<td>936</td>
</tr>
<tr>
<td>GETSOL</td>
<td>937</td>
</tr>
<tr>
<td>GI</td>
<td>938</td>
</tr>
<tr>
<td>GIAI</td>
<td>939</td>
</tr>
<tr>
<td>GIMODEL</td>
<td>940</td>
</tr>
<tr>
<td>GINJGAS</td>
<td>941</td>
</tr>
<tr>
<td>GINODE</td>
<td>943</td>
</tr>
<tr>
<td>GLIFTLIM</td>
<td>944</td>
</tr>
<tr>
<td>GLIFTOPT</td>
<td>945</td>
</tr>
<tr>
<td>GLTHEX</td>
<td>946</td>
</tr>
<tr>
<td>GMDISBC</td>
<td>947</td>
</tr>
<tr>
<td>GMDISBCL</td>
<td>949</td>
</tr>
<tr>
<td>GMPSTBC</td>
<td>951</td>
</tr>
<tr>
<td>GMPSTBCL</td>
<td>954</td>
</tr>
<tr>
<td>GMTRABC</td>
<td>957</td>
</tr>
<tr>
<td>GMTRABCL</td>
<td>959</td>
</tr>
<tr>
<td>GNETDP</td>
<td>961</td>
</tr>
<tr>
<td>GNETINJE</td>
<td>963</td>
</tr>
<tr>
<td>GNETPUMP</td>
<td>964</td>
</tr>
<tr>
<td>GPMaint</td>
<td>966</td>
</tr>
<tr>
<td>GPMaint3</td>
<td>970</td>
</tr>
<tr>
<td>GPTABLE</td>
<td>974</td>
</tr>
<tr>
<td>GPTABLEN</td>
<td>975</td>
</tr>
<tr>
<td>GPTABLE3</td>
<td>976</td>
</tr>
<tr>
<td>GPTDIMS</td>
<td>977</td>
</tr>
<tr>
<td>GQUALITY</td>
<td>978</td>
</tr>
<tr>
<td>GRADGROUP</td>
<td>979</td>
</tr>
<tr>
<td>GRADRESV</td>
<td>980</td>
</tr>
<tr>
<td>GRADRFT</td>
<td>983</td>
</tr>
<tr>
<td>GRADWELL</td>
<td>984</td>
</tr>
<tr>
<td>GRAVCONS</td>
<td>985</td>
</tr>
<tr>
<td>GRAVDR</td>
<td>986</td>
</tr>
<tr>
<td>GRAVDRB</td>
<td>987</td>
</tr>
<tr>
<td>Topic</td>
<td>Page</td>
</tr>
<tr>
<td>---------------------</td>
<td>------</td>
</tr>
<tr>
<td>ECLIPSE Reference Manual</td>
<td></td>
</tr>
<tr>
<td>GRAVDRM</td>
<td>988</td>
</tr>
<tr>
<td>GRAVITY</td>
<td>989</td>
</tr>
<tr>
<td>GRDREACH</td>
<td>990</td>
</tr>
<tr>
<td>GREF</td>
<td>993</td>
</tr>
<tr>
<td>GREFS</td>
<td>994</td>
</tr>
<tr>
<td>GRIDFILE</td>
<td>995</td>
</tr>
<tr>
<td>Examples</td>
<td>996</td>
</tr>
<tr>
<td>GRIDOPTS</td>
<td>997</td>
</tr>
<tr>
<td>GRIDUNIT</td>
<td>999</td>
</tr>
<tr>
<td>GRUPFUEL</td>
<td>1000</td>
</tr>
<tr>
<td>GRUPGR</td>
<td>1002</td>
</tr>
<tr>
<td>GRUPINJE</td>
<td>1004</td>
</tr>
<tr>
<td>Examples</td>
<td>1007</td>
</tr>
<tr>
<td>GRUPLIM</td>
<td>1009</td>
</tr>
<tr>
<td>GRUPMAST</td>
<td>1010</td>
</tr>
<tr>
<td>GRUPNET</td>
<td>1012</td>
</tr>
<tr>
<td>GRUPPROD</td>
<td>1015</td>
</tr>
<tr>
<td>GRUPRIG</td>
<td>1017</td>
</tr>
<tr>
<td>Examples</td>
<td>1019</td>
</tr>
<tr>
<td>GRUPSALE</td>
<td>1020</td>
</tr>
<tr>
<td>GRUPSLAV</td>
<td>1022</td>
</tr>
<tr>
<td>GRUPTARG</td>
<td>1025</td>
</tr>
<tr>
<td>GRUPTREE</td>
<td>1027</td>
</tr>
<tr>
<td>GSATCOMP</td>
<td>1029</td>
</tr>
<tr>
<td>GSATINJE</td>
<td>1030</td>
</tr>
<tr>
<td>GSATPROD</td>
<td>1032</td>
</tr>
<tr>
<td>GSEPCOND</td>
<td>1035</td>
</tr>
<tr>
<td>Examples</td>
<td>1035</td>
</tr>
<tr>
<td>GSEPREPT</td>
<td>1036</td>
</tr>
<tr>
<td>GSF</td>
<td>1038</td>
</tr>
<tr>
<td>GSSCPTST</td>
<td>1039</td>
</tr>
<tr>
<td>GSWINGF</td>
<td>1042</td>
</tr>
<tr>
<td>GTADD</td>
<td>1044</td>
</tr>
<tr>
<td>GTMULT</td>
<td>1046</td>
</tr>
<tr>
<td>GUIDECAL</td>
<td>1048</td>
</tr>
<tr>
<td>GUIDERAT</td>
<td>1049</td>
</tr>
<tr>
<td>Examples</td>
<td>1052</td>
</tr>
<tr>
<td>GUPFREQ</td>
<td>1053</td>
</tr>
<tr>
<td>GWRTWCV</td>
<td>1054</td>
</tr>
</tbody>
</table>

**H** ................................................................. 1055

H2SSOL ......................................................... 1056
HALFTRAN ...................................................... 1057
HAxxxxxx ...................................................... 1058
HDISP ......................................................... 1059
HEATCR ......................................................... 1060
<table>
<thead>
<tr>
<th>Command</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>HSWRG</td>
<td>1116</td>
</tr>
<tr>
<td>HSWRO</td>
<td>1117</td>
</tr>
<tr>
<td>HWELLS</td>
<td>1118</td>
</tr>
<tr>
<td>HKWRO</td>
<td>1119</td>
</tr>
<tr>
<td>HKKROG</td>
<td>1120</td>
</tr>
<tr>
<td>HKKORW</td>
<td>1121</td>
</tr>
<tr>
<td>HKR RW</td>
<td>1122</td>
</tr>
<tr>
<td>HKR WR</td>
<td>1123</td>
</tr>
<tr>
<td>HWPCW</td>
<td>1124</td>
</tr>
<tr>
<td>HWSNUM</td>
<td>1125</td>
</tr>
<tr>
<td>HWSONGCR</td>
<td>1126</td>
</tr>
<tr>
<td>HWSOWCR</td>
<td>1127</td>
</tr>
<tr>
<td>HWSWC R</td>
<td>1128</td>
</tr>
<tr>
<td>HWSWL</td>
<td>1129</td>
</tr>
<tr>
<td>HWSWLPC</td>
<td>1130</td>
</tr>
<tr>
<td>HWSWU</td>
<td>1131</td>
</tr>
<tr>
<td>HXFIN</td>
<td>1132</td>
</tr>
<tr>
<td>HYDRHEAD</td>
<td>1133</td>
</tr>
<tr>
<td>HYDRO</td>
<td>1134</td>
</tr>
<tr>
<td>HYFIN</td>
<td>1135</td>
</tr>
<tr>
<td>HYKR</td>
<td>1136</td>
</tr>
<tr>
<td>HYMOBGDR</td>
<td>1137</td>
</tr>
<tr>
<td>HYP C</td>
<td>1138</td>
</tr>
<tr>
<td>HYST</td>
<td>1139</td>
</tr>
<tr>
<td>HYSTCHCK</td>
<td>1140</td>
</tr>
<tr>
<td>HYST J</td>
<td>1141</td>
</tr>
<tr>
<td>HYST K</td>
<td>1142</td>
</tr>
<tr>
<td>HZFIN</td>
<td>1143</td>
</tr>
<tr>
<td>IHOST</td>
<td>1144</td>
</tr>
<tr>
<td>IKR G-ORW</td>
<td>1145</td>
</tr>
<tr>
<td>IKU3P</td>
<td>1146</td>
</tr>
<tr>
<td>IMBNUM</td>
<td>1147</td>
</tr>
<tr>
<td>IMBNUM MF</td>
<td>1148</td>
</tr>
<tr>
<td>IMKRD VD</td>
<td>1149</td>
</tr>
<tr>
<td>IMP CVD</td>
<td>1150</td>
</tr>
<tr>
<td>IMPES</td>
<td>1151</td>
</tr>
<tr>
<td>IMPFILE</td>
<td>1152</td>
</tr>
<tr>
<td>IMPLICIT</td>
<td>1153</td>
</tr>
<tr>
<td>IMPORT</td>
<td>1154</td>
</tr>
<tr>
<td>IMP SAT</td>
<td>1155</td>
</tr>
<tr>
<td>IMP TVD</td>
<td>1156</td>
</tr>
<tr>
<td>IMSPCVD</td>
<td>1157</td>
</tr>
<tr>
<td>INCLUDE</td>
<td>1158</td>
</tr>
<tr>
<td>Examples</td>
<td>1159</td>
</tr>
<tr>
<td>HXFIN</td>
<td>1160</td>
</tr>
</tbody>
</table>

*Examples* 1162
<table>
<thead>
<tr>
<th>K</th>
<th>L</th>
</tr>
</thead>
<tbody>
<tr>
<td>INIT</td>
<td>LAB</td>
</tr>
<tr>
<td>INRAD</td>
<td></td>
</tr>
<tr>
<td>INSPEC</td>
<td></td>
</tr>
<tr>
<td>INTPC</td>
<td></td>
</tr>
<tr>
<td>IONROCK</td>
<td></td>
</tr>
<tr>
<td>IONXROCK</td>
<td></td>
</tr>
<tr>
<td>IONXSURF</td>
<td></td>
</tr>
<tr>
<td>IPCW,IPCG</td>
<td></td>
</tr>
<tr>
<td>ISGAS</td>
<td></td>
</tr>
<tr>
<td>ISTNUM</td>
<td></td>
</tr>
<tr>
<td>ISGL-OWCR</td>
<td></td>
</tr>
<tr>
<td>ISOLNUM</td>
<td></td>
</tr>
<tr>
<td></td>
<td>J</td>
</tr>
<tr>
<td>JALS</td>
<td></td>
</tr>
<tr>
<td>JFUNC</td>
<td></td>
</tr>
<tr>
<td>JFUNCIR</td>
<td></td>
</tr>
<tr>
<td></td>
<td>K</td>
</tr>
<tr>
<td>KRGDI</td>
<td></td>
</tr>
<tr>
<td>KRGDM</td>
<td></td>
</tr>
<tr>
<td>KRG-IKRGR</td>
<td></td>
</tr>
<tr>
<td>KRNUM</td>
<td></td>
</tr>
<tr>
<td>KRNUMMF</td>
<td></td>
</tr>
<tr>
<td>KRODI</td>
<td></td>
</tr>
<tr>
<td>KRODM</td>
<td></td>
</tr>
<tr>
<td>KRO-ORG</td>
<td></td>
</tr>
<tr>
<td>KRWDI</td>
<td></td>
</tr>
<tr>
<td>KRWDM</td>
<td></td>
</tr>
<tr>
<td>KRW-IKRW</td>
<td></td>
</tr>
<tr>
<td>KVALUES</td>
<td></td>
</tr>
<tr>
<td>KVAN</td>
<td></td>
</tr>
<tr>
<td>KVCMP</td>
<td></td>
</tr>
<tr>
<td>KVCR</td>
<td></td>
</tr>
<tr>
<td>KVCRS</td>
<td></td>
</tr>
<tr>
<td>KVCRWAT</td>
<td></td>
</tr>
<tr>
<td>KVGEN</td>
<td></td>
</tr>
<tr>
<td>KVMF</td>
<td></td>
</tr>
<tr>
<td>KVTCBCn</td>
<td></td>
</tr>
<tr>
<td>KVTABLE</td>
<td></td>
</tr>
<tr>
<td>KVTABTn</td>
<td></td>
</tr>
<tr>
<td>KVTEMP</td>
<td></td>
</tr>
<tr>
<td>KVVII</td>
<td></td>
</tr>
<tr>
<td>KWTABTn</td>
<td></td>
</tr>
<tr>
<td>KWTEMP</td>
<td></td>
</tr>
<tr>
<td></td>
<td>L</td>
</tr>
<tr>
<td>LAB</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Command</td>
<td>Page</td>
</tr>
<tr>
<td>--------------</td>
<td>------</td>
</tr>
<tr>
<td>MAPAXES</td>
<td>1302</td>
</tr>
<tr>
<td>MAPUNITS</td>
<td>1304</td>
</tr>
<tr>
<td>MASSFLOW</td>
<td>1305</td>
</tr>
<tr>
<td>MATCORR</td>
<td>1306</td>
</tr>
<tr>
<td>Examples</td>
<td></td>
</tr>
<tr>
<td>MAXTRANZ</td>
<td>1307</td>
</tr>
<tr>
<td>MAXVALUE</td>
<td>1308</td>
</tr>
<tr>
<td>MEMORY</td>
<td>1309</td>
</tr>
<tr>
<td>MEMSAVE</td>
<td>1310</td>
</tr>
<tr>
<td>MESSAGE</td>
<td>1311</td>
</tr>
<tr>
<td>MESSAGES</td>
<td>1312</td>
</tr>
<tr>
<td>MESSOPTS</td>
<td>1314</td>
</tr>
<tr>
<td>METRIC</td>
<td>1315</td>
</tr>
<tr>
<td>MIDS</td>
<td>1316</td>
</tr>
<tr>
<td>MINDZNET</td>
<td>1317</td>
</tr>
<tr>
<td>MINNNCT</td>
<td>1318</td>
</tr>
<tr>
<td>MINPORV</td>
<td>1319</td>
</tr>
<tr>
<td>MINPORV9</td>
<td>1320</td>
</tr>
<tr>
<td>MINPV</td>
<td>1321</td>
</tr>
<tr>
<td>MINPVV</td>
<td>1322</td>
</tr>
<tr>
<td>LSWL</td>
<td>1276</td>
</tr>
<tr>
<td>LSWLPC</td>
<td>1277</td>
</tr>
<tr>
<td>LSWU</td>
<td>1278</td>
</tr>
<tr>
<td>LTSIGMA</td>
<td>1279</td>
</tr>
<tr>
<td>LTRACE</td>
<td>1281</td>
</tr>
<tr>
<td>LUMPINGS</td>
<td>1282</td>
</tr>
<tr>
<td>LUMPING</td>
<td>1283</td>
</tr>
<tr>
<td>LWKRO</td>
<td>1284</td>
</tr>
<tr>
<td>LWKROR</td>
<td>1285</td>
</tr>
<tr>
<td>LWKRORW</td>
<td>1286</td>
</tr>
<tr>
<td>LWKRW</td>
<td>1287</td>
</tr>
<tr>
<td>LWKRWR</td>
<td>1288</td>
</tr>
<tr>
<td>LWPCW</td>
<td>1289</td>
</tr>
<tr>
<td>LWSLTS</td>
<td>1290</td>
</tr>
<tr>
<td>LWSNUM</td>
<td>1291</td>
</tr>
<tr>
<td>LWSOGR</td>
<td>1292</td>
</tr>
<tr>
<td>LWSOWCR</td>
<td>1293</td>
</tr>
<tr>
<td>LWSWCR</td>
<td>1294</td>
</tr>
<tr>
<td>LWSWL</td>
<td>1295</td>
</tr>
<tr>
<td>LWSWLPC</td>
<td>1296</td>
</tr>
<tr>
<td>LWSWU</td>
<td>1297</td>
</tr>
<tr>
<td>LX</td>
<td>1298</td>
</tr>
<tr>
<td>LY</td>
<td>1299</td>
</tr>
<tr>
<td>LZ</td>
<td>1300</td>
</tr>
<tr>
<td>Topic</td>
<td>Page</td>
</tr>
<tr>
<td>------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>MINROCKV</td>
<td>1323</td>
</tr>
<tr>
<td>MINRV</td>
<td>1324</td>
</tr>
<tr>
<td>MINVALUE</td>
<td>1325</td>
</tr>
<tr>
<td>MISC</td>
<td>1326</td>
</tr>
<tr>
<td>MISCEXP</td>
<td>1327</td>
</tr>
<tr>
<td>MISCIBLE</td>
<td>1328</td>
</tr>
<tr>
<td>Enable the miscible gas flood option (ECLIPSE 100)</td>
<td>1328</td>
</tr>
<tr>
<td>Use dependence on surface tensions in properties (ECLIPSE 300)</td>
<td>1329</td>
</tr>
<tr>
<td>MISCNUM</td>
<td>1330</td>
</tr>
<tr>
<td>ECLIPSE 100 miscible flood model</td>
<td>1330</td>
</tr>
<tr>
<td>ECLIPSE 300 miscible model - surface tension effects</td>
<td>1330</td>
</tr>
<tr>
<td>MISCSTR</td>
<td>1332</td>
</tr>
<tr>
<td>MISCSTRP</td>
<td>1333</td>
</tr>
<tr>
<td>MISCSTRP</td>
<td>1334</td>
</tr>
<tr>
<td>MIXACTG</td>
<td>1336</td>
</tr>
<tr>
<td>MIXACTO</td>
<td>1338</td>
</tr>
<tr>
<td>MIXWPERM</td>
<td>1340</td>
</tr>
<tr>
<td>MLANG</td>
<td>1341</td>
</tr>
<tr>
<td>MLANGLSV</td>
<td>1342</td>
</tr>
<tr>
<td>MONITOR</td>
<td>1343</td>
</tr>
<tr>
<td>MPFA</td>
<td>1344</td>
</tr>
<tr>
<td>MPFANUM</td>
<td>1347</td>
</tr>
<tr>
<td>MPFNNC</td>
<td>1348</td>
</tr>
<tr>
<td>MSFN</td>
<td>1350</td>
</tr>
<tr>
<td>MSGFILE</td>
<td>1352</td>
</tr>
<tr>
<td>MULSGGD</td>
<td>1353</td>
</tr>
<tr>
<td>MULSGGDV</td>
<td>1354</td>
</tr>
<tr>
<td>MULTFLT</td>
<td>1355</td>
</tr>
<tr>
<td>MULTIN</td>
<td>1357</td>
</tr>
<tr>
<td>MULTIN</td>
<td>1358</td>
</tr>
<tr>
<td>MULTIPHA</td>
<td>1359</td>
</tr>
<tr>
<td>MULTIPLY</td>
<td>1359</td>
</tr>
<tr>
<td>Legal arrays</td>
<td>1360</td>
</tr>
<tr>
<td>Notes</td>
<td>1361</td>
</tr>
<tr>
<td>Examples</td>
<td>1361</td>
</tr>
<tr>
<td>MULTIREG</td>
<td>1363</td>
</tr>
<tr>
<td>Legal arrays</td>
<td>1363</td>
</tr>
<tr>
<td>Notes</td>
<td>1365</td>
</tr>
<tr>
<td>Additional notes</td>
<td>1365</td>
</tr>
<tr>
<td>Examples</td>
<td>1366</td>
</tr>
<tr>
<td>MULTMF</td>
<td>1367</td>
</tr>
<tr>
<td>MULTNUM</td>
<td>1368</td>
</tr>
<tr>
<td>MULTOUT</td>
<td>1369</td>
</tr>
<tr>
<td>MULTOUTS</td>
<td>1370</td>
</tr>
<tr>
<td>MULTPV</td>
<td>1371</td>
</tr>
<tr>
<td>MULTTR</td>
<td>1372</td>
</tr>
<tr>
<td>Command</td>
<td>Page</td>
</tr>
<tr>
<td>---------------</td>
<td>------</td>
</tr>
<tr>
<td>MULTREAL</td>
<td>1374</td>
</tr>
<tr>
<td>MULTREGD</td>
<td>1375</td>
</tr>
<tr>
<td>MULTREGH</td>
<td>1377</td>
</tr>
<tr>
<td>MULTREGP</td>
<td>1379</td>
</tr>
<tr>
<td>MULTREGT</td>
<td>1381</td>
</tr>
<tr>
<td>MULTSAVE</td>
<td>1384</td>
</tr>
<tr>
<td>MULTSIG</td>
<td>1385</td>
</tr>
<tr>
<td>MULTSIGV</td>
<td>1386</td>
</tr>
<tr>
<td>MULTTHT</td>
<td>1387</td>
</tr>
<tr>
<td>MULTTHT-</td>
<td>1388</td>
</tr>
<tr>
<td>MULTX</td>
<td>1389</td>
</tr>
<tr>
<td>MULTX-</td>
<td>1391</td>
</tr>
<tr>
<td>MULTY</td>
<td>1393</td>
</tr>
<tr>
<td>MULTY-</td>
<td>1395</td>
</tr>
<tr>
<td>MULTZ</td>
<td>1397</td>
</tr>
<tr>
<td>MULTZ-</td>
<td>1399</td>
</tr>
<tr>
<td>MW</td>
<td>1401</td>
</tr>
<tr>
<td>MWDETAIL</td>
<td>1402</td>
</tr>
<tr>
<td>MWS</td>
<td>1403</td>
</tr>
<tr>
<td>MWWW</td>
<td>1404</td>
</tr>
<tr>
<td>MWWS</td>
<td>1405</td>
</tr>
<tr>
<td>NARROW</td>
<td>1407</td>
</tr>
<tr>
<td>NCOMPS</td>
<td>1408</td>
</tr>
<tr>
<td>NCONSUMP</td>
<td>1409</td>
</tr>
<tr>
<td>NEFAC</td>
<td>1410</td>
</tr>
<tr>
<td>NEI</td>
<td>1411</td>
</tr>
<tr>
<td>NETBALAN</td>
<td>1412</td>
</tr>
<tr>
<td>NETCOMPA</td>
<td>1414</td>
</tr>
<tr>
<td>NETWORK</td>
<td>1415</td>
</tr>
<tr>
<td>NEWTON</td>
<td>1418</td>
</tr>
<tr>
<td>NEWTRAN</td>
<td>1419</td>
</tr>
<tr>
<td>NEXT</td>
<td>1420</td>
</tr>
<tr>
<td>NEXTSTEP</td>
<td>1421</td>
</tr>
<tr>
<td>NEXTSTPL</td>
<td>1422</td>
</tr>
<tr>
<td>NGASREM</td>
<td>1423</td>
</tr>
<tr>
<td>NGASREM</td>
<td>1424</td>
</tr>
<tr>
<td>NINEC</td>
<td>1426</td>
</tr>
<tr>
<td>NINEPOIN</td>
<td>1427</td>
</tr>
<tr>
<td>NINEXZ</td>
<td>1428</td>
</tr>
<tr>
<td>NINEYX</td>
<td>1429</td>
</tr>
<tr>
<td>NMATOPTS</td>
<td>1430</td>
</tr>
<tr>
<td>NMATRIX</td>
<td>1432</td>
</tr>
<tr>
<td>Keywords</td>
<td>Pages</td>
</tr>
<tr>
<td>--------------------------</td>
<td>-------</td>
</tr>
<tr>
<td>OILAPI</td>
<td>1482</td>
</tr>
<tr>
<td>OILCOMPR</td>
<td>1483</td>
</tr>
<tr>
<td>OILMW</td>
<td>1484</td>
</tr>
<tr>
<td>OILSPECH</td>
<td>1485</td>
</tr>
<tr>
<td>OILVINDX</td>
<td>1486</td>
</tr>
<tr>
<td>OILVISCC</td>
<td>1488</td>
</tr>
<tr>
<td>OILVISCF</td>
<td>1492</td>
</tr>
<tr>
<td>OILVISCT</td>
<td>1494</td>
</tr>
<tr>
<td>OILVTIM</td>
<td>1497</td>
</tr>
<tr>
<td>OLDEQUIL</td>
<td>1498</td>
</tr>
<tr>
<td>OLDTRAN</td>
<td>1499</td>
</tr>
<tr>
<td>OLDTRANR</td>
<td>1500</td>
</tr>
<tr>
<td>OMEGAA</td>
<td>1501</td>
</tr>
<tr>
<td>OMEGAADE</td>
<td>1502</td>
</tr>
<tr>
<td>OMEGAAAS</td>
<td>1503</td>
</tr>
<tr>
<td>OMEGAAASD</td>
<td>1504</td>
</tr>
<tr>
<td>OMEGAB</td>
<td>1505</td>
</tr>
<tr>
<td>OMEGABDE</td>
<td>1506</td>
</tr>
<tr>
<td>OMEGABS</td>
<td>1507</td>
</tr>
<tr>
<td>OMEGABSD</td>
<td>1508</td>
</tr>
<tr>
<td>OPERATE</td>
<td>1509</td>
</tr>
<tr>
<td>Legal arrays</td>
<td>1510</td>
</tr>
<tr>
<td>Notes</td>
<td>1511</td>
</tr>
<tr>
<td>Examples</td>
<td>1512</td>
</tr>
<tr>
<td>OPERATER</td>
<td>1514</td>
</tr>
<tr>
<td>Legal arrays</td>
<td>1515</td>
</tr>
<tr>
<td>Notes</td>
<td>1516</td>
</tr>
<tr>
<td>Additional notes</td>
<td>1517</td>
</tr>
<tr>
<td>Examples</td>
<td>1518</td>
</tr>
<tr>
<td>OPERNUM</td>
<td>1520</td>
</tr>
<tr>
<td>OPTCONS</td>
<td>1521</td>
</tr>
<tr>
<td>OPTDIMS</td>
<td>1523</td>
</tr>
<tr>
<td>OPTFUNC</td>
<td>1524</td>
</tr>
<tr>
<td>OPTIONS</td>
<td>1526</td>
</tr>
<tr>
<td>Examples</td>
<td>1548</td>
</tr>
<tr>
<td>OPTIONS3</td>
<td>1549</td>
</tr>
<tr>
<td>Examples</td>
<td>1580</td>
</tr>
<tr>
<td>OPTLOAD</td>
<td>1581</td>
</tr>
<tr>
<td>OPTOBD3P</td>
<td>1582</td>
</tr>
<tr>
<td>OPTOPTS</td>
<td>1584</td>
</tr>
<tr>
<td>OPTPARS</td>
<td>1585</td>
</tr>
<tr>
<td>Examples</td>
<td>1587</td>
</tr>
<tr>
<td>OPTTUNE</td>
<td>1588</td>
</tr>
<tr>
<td>ORTHERERO</td>
<td>1589</td>
</tr>
<tr>
<td>Symbol</td>
<td>Page</td>
</tr>
<tr>
<td>------------</td>
<td>------</td>
</tr>
<tr>
<td>PMANUM</td>
<td>1703</td>
</tr>
<tr>
<td>PMAX</td>
<td>1704</td>
</tr>
<tr>
<td>PMISC</td>
<td>1706</td>
</tr>
<tr>
<td>POISSONR</td>
<td>1707</td>
</tr>
<tr>
<td>POLYMER</td>
<td>1708</td>
</tr>
<tr>
<td>PORO</td>
<td>1709</td>
</tr>
<tr>
<td>PORV</td>
<td>1710</td>
</tr>
<tr>
<td>PPCWMAX</td>
<td>1711</td>
</tr>
<tr>
<td>PRCORR</td>
<td>1712</td>
</tr>
<tr>
<td>PREF</td>
<td>1713</td>
</tr>
<tr>
<td>PREFS</td>
<td>1714</td>
</tr>
<tr>
<td>PREFT</td>
<td>1715</td>
</tr>
<tr>
<td>PREFTS</td>
<td>1716</td>
</tr>
<tr>
<td>PREFW</td>
<td>1717</td>
</tr>
<tr>
<td>PREFWS</td>
<td>1718</td>
</tr>
<tr>
<td>PRESSURE</td>
<td>1719</td>
</tr>
<tr>
<td>PRIORITY</td>
<td>1720</td>
</tr>
<tr>
<td>PREFTS</td>
<td>1722</td>
</tr>
<tr>
<td>PROPFLOW</td>
<td>1723</td>
</tr>
<tr>
<td>PRORDER</td>
<td>1724</td>
</tr>
<tr>
<td>PRVD</td>
<td>1728</td>
</tr>
<tr>
<td>PSEUPRES</td>
<td>1729</td>
</tr>
<tr>
<td>PSGRO</td>
<td>1730</td>
</tr>
<tr>
<td>PSGRW</td>
<td>1731</td>
</tr>
<tr>
<td>PSGRWS</td>
<td>1732</td>
</tr>
<tr>
<td>PSORW</td>
<td>1733</td>
</tr>
<tr>
<td>PSPLITX</td>
<td>1734</td>
</tr>
<tr>
<td>PSPLITY</td>
<td>1735</td>
</tr>
<tr>
<td>PSSTA</td>
<td>1736</td>
</tr>
<tr>
<td>PSTEADY</td>
<td>1737</td>
</tr>
<tr>
<td>PSTNUM</td>
<td>1739</td>
</tr>
<tr>
<td>PSTRBC</td>
<td>1740</td>
</tr>
<tr>
<td>PSTRBCCL</td>
<td>1743</td>
</tr>
<tr>
<td>PSWG</td>
<td>1744</td>
</tr>
<tr>
<td>PSWRO</td>
<td>1747</td>
</tr>
<tr>
<td>PVCDO</td>
<td>1748</td>
</tr>
<tr>
<td>PVCO</td>
<td>1750</td>
</tr>
<tr>
<td>PVDG</td>
<td>1753</td>
</tr>
<tr>
<td>PVDO</td>
<td>1754</td>
</tr>
<tr>
<td>PVDS</td>
<td>1755</td>
</tr>
<tr>
<td>PVTG</td>
<td>1756</td>
</tr>
<tr>
<td>PVT-M</td>
<td>1758</td>
</tr>
<tr>
<td>PVTNUM</td>
<td>1759</td>
</tr>
<tr>
<td>PVTO</td>
<td>1760</td>
</tr>
</tbody>
</table>
ECLIPSE Reference Manual

PVTW ................................................................. 1762
PVTWSALT ...................................................... 1764
PVZG ............................................................... 1767

Q ................................................................. 1769
QDRILL ............................................................. 1770
QHRATING ....................................................... 1771
QMOBIL ........................................................... 1772

R ................................................................. 1773
RADFIN4 .......................................................... 1774
RADFIN ............................................................ 1776
RADIAL ............................................................. 1779
RAINFALL ......................................................... 1780
RBEDCONT ....................................................... 1783
RCMASTS ......................................................... 1784
REACACT .......................................................... 1785
RE ACCORD ........................................................ 1786
RE ACCRIT ........................................................ 1787
RE ACENTH ....................................................... 1788
RE ACES ............................................................ 1789
RE A CLIMS ....................................................... 1792
RE ACOPS .......................................................... 1793
RE AC PHA ........................................................ 1794
RE AC PORD ...................................................... 1795
RE AC RATE ....................................................... 1796
RE AC SM AX ..................................................... 1797
RE AC SORD ....................................................... 1798
REA CTION ......................................................... 1799
REA CTRAC ......................................................... 1800
READDATA ....................................................... 1801
RECOVERY ........................................................ 1802
REFINE ............................................................ 1803

Examples ....................................................... 1806
REGDIMS .......................................................... 1808
RESIDNUM ....................................................... 1810
RES OPT ............................................................. 1811
RES ORB ............................................................ 1812
RESTART ........................................................... 1813
RES VNUM .......................................................... 1814
RIVDEBUG ........................................................ 1815
RIVERSYS ........................................................ 1816

Examples ....................................................... 1817
RIVRDIMS ........................................................ 1818
RIVRPROP ........................................................ 1820
RIVRXSEC ....................................................... 1821
RIVSALT ........................................................... 1822
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>RIVTRACE</td>
<td>1823</td>
</tr>
<tr>
<td>RKPMNMX</td>
<td>1825</td>
</tr>
<tr>
<td>RKTREMDIR</td>
<td>1826</td>
</tr>
<tr>
<td>ROCK</td>
<td>1827</td>
</tr>
<tr>
<td>ROCK2D</td>
<td>1830</td>
</tr>
<tr>
<td>ROCK2DTR</td>
<td>1832</td>
</tr>
<tr>
<td>ROCKCOMP</td>
<td>1834</td>
</tr>
<tr>
<td>ROCKCON</td>
<td>1836</td>
</tr>
<tr>
<td>ROCKCONL</td>
<td>1838</td>
</tr>
<tr>
<td>ROCKDEN</td>
<td>1840</td>
</tr>
<tr>
<td>ROCKDIMS</td>
<td>1841</td>
</tr>
<tr>
<td>ROCKFRAC</td>
<td>1842</td>
</tr>
<tr>
<td>ROCKNUM</td>
<td>1843</td>
</tr>
<tr>
<td>ROCKOPTS</td>
<td>1844</td>
</tr>
<tr>
<td>ROCKPAMA</td>
<td>1846</td>
</tr>
<tr>
<td>ROCKPAME</td>
<td>1851</td>
</tr>
<tr>
<td>ROCKPMAT</td>
<td>1854</td>
</tr>
<tr>
<td>ROCKPROP</td>
<td>1856</td>
</tr>
<tr>
<td>ROCKSPLV</td>
<td>1857</td>
</tr>
<tr>
<td>ROCKTAB</td>
<td>1858</td>
</tr>
<tr>
<td>ROCKTABH</td>
<td>1860</td>
</tr>
<tr>
<td>ROCKTABW</td>
<td>1863</td>
</tr>
<tr>
<td>ROCKTHSG</td>
<td>1864</td>
</tr>
<tr>
<td>ROCKTRMX</td>
<td>1867</td>
</tr>
<tr>
<td>ROCKTRMY</td>
<td>1869</td>
</tr>
<tr>
<td>ROCKTRMZ</td>
<td>1871</td>
</tr>
<tr>
<td>ROCKTSIG</td>
<td>1873</td>
</tr>
<tr>
<td>ROCKV</td>
<td>1875</td>
</tr>
<tr>
<td>ROCKWNOD</td>
<td>1876</td>
</tr>
<tr>
<td>ROMF</td>
<td>1878</td>
</tr>
<tr>
<td>RPRKPAME</td>
<td>1879</td>
</tr>
<tr>
<td>RPTGRID</td>
<td>1880</td>
</tr>
<tr>
<td>Examples</td>
<td>1884</td>
</tr>
<tr>
<td>RPTGRIDL</td>
<td>1885</td>
</tr>
<tr>
<td>RPTHMD</td>
<td>1886</td>
</tr>
<tr>
<td>RPTHMG</td>
<td>1887</td>
</tr>
<tr>
<td>Examples</td>
<td>1887</td>
</tr>
<tr>
<td>RPTHMW</td>
<td>1888</td>
</tr>
<tr>
<td>Examples</td>
<td>1888</td>
</tr>
<tr>
<td>RPTINIT</td>
<td>1890</td>
</tr>
<tr>
<td>RPTISOL</td>
<td>1893</td>
</tr>
<tr>
<td>RPTONLY</td>
<td>1894</td>
</tr>
<tr>
<td>RPTONLYO</td>
<td>1895</td>
</tr>
<tr>
<td>RPTOPT</td>
<td>1896</td>
</tr>
<tr>
<td>RPTPRINT</td>
<td>1897</td>
</tr>
<tr>
<td>Module</td>
<td>Pages</td>
</tr>
<tr>
<td>------------</td>
<td>---------</td>
</tr>
<tr>
<td>RPTPROPS</td>
<td>1899</td>
</tr>
<tr>
<td>RPTREGS</td>
<td>1903</td>
</tr>
<tr>
<td>RPTTRST</td>
<td>1908</td>
</tr>
<tr>
<td>RPTTRUST</td>
<td>1927</td>
</tr>
<tr>
<td>RPTSLN</td>
<td>1946</td>
</tr>
<tr>
<td>RPTSOL</td>
<td>1948</td>
</tr>
<tr>
<td>RS</td>
<td>1963</td>
</tr>
<tr>
<td>RSCONST</td>
<td>1964</td>
</tr>
<tr>
<td>RSCONSTT</td>
<td>1965</td>
</tr>
<tr>
<td>RSGI</td>
<td>1966</td>
</tr>
<tr>
<td>RSM</td>
<td>1967</td>
</tr>
<tr>
<td>RSSPEC</td>
<td>1968</td>
</tr>
<tr>
<td>RSVS</td>
<td>1969</td>
</tr>
<tr>
<td>RSW</td>
<td>1970</td>
</tr>
<tr>
<td>RSWVD</td>
<td>1971</td>
</tr>
<tr>
<td>RTEMP</td>
<td>1973</td>
</tr>
<tr>
<td>RTEMPC</td>
<td>1974</td>
</tr>
<tr>
<td>RTEMPV</td>
<td>1975</td>
</tr>
<tr>
<td>RUNSUM</td>
<td>1976</td>
</tr>
<tr>
<td>RV</td>
<td>1977</td>
</tr>
<tr>
<td>RVCONST</td>
<td>1978</td>
</tr>
<tr>
<td>RVCONSTT</td>
<td>1979</td>
</tr>
<tr>
<td>RVGI</td>
<td>1980</td>
</tr>
<tr>
<td>RVVD</td>
<td>1981</td>
</tr>
<tr>
<td>SALT</td>
<td>1982</td>
</tr>
<tr>
<td>SALNITR</td>
<td>1983</td>
</tr>
<tr>
<td>SALNITY</td>
<td>1984</td>
</tr>
<tr>
<td>SALT</td>
<td>1985</td>
</tr>
<tr>
<td>SALTEFF</td>
<td>1986</td>
</tr>
<tr>
<td>SALTNODE</td>
<td>1988</td>
</tr>
<tr>
<td>SALTVD</td>
<td>1989</td>
</tr>
<tr>
<td>SAMG</td>
<td>1990</td>
</tr>
<tr>
<td>SATNUM</td>
<td>1991</td>
</tr>
<tr>
<td>Command</td>
<td>Page</td>
</tr>
<tr>
<td>---------</td>
<td>------</td>
</tr>
<tr>
<td>SATOPTS</td>
<td>1992</td>
</tr>
<tr>
<td>SAVE</td>
<td>1994</td>
</tr>
<tr>
<td></td>
<td>1994</td>
</tr>
<tr>
<td>SAVEEND</td>
<td>1996</td>
</tr>
<tr>
<td>SCALECRS</td>
<td>1997</td>
</tr>
<tr>
<td>SCALELIM</td>
<td>1999</td>
</tr>
<tr>
<td>SCDATA</td>
<td>2000</td>
</tr>
<tr>
<td>SCDATABG</td>
<td>2001</td>
</tr>
<tr>
<td>SCDETAB</td>
<td>2002</td>
</tr>
<tr>
<td>SCDPDIMS</td>
<td>2004</td>
</tr>
<tr>
<td>SCDPTAB</td>
<td>2005</td>
</tr>
<tr>
<td>SCDPTRAC</td>
<td>2006</td>
</tr>
<tr>
<td>SCFDIMS</td>
<td>2007</td>
</tr>
<tr>
<td>SCONINJE</td>
<td>2008</td>
</tr>
<tr>
<td>SCONPROD</td>
<td>2010</td>
</tr>
<tr>
<td>SCREF</td>
<td>2013</td>
</tr>
<tr>
<td>SCVD</td>
<td>2014</td>
</tr>
<tr>
<td>SDENSITY</td>
<td>2015</td>
</tr>
<tr>
<td>SDREF</td>
<td>2016</td>
</tr>
<tr>
<td></td>
<td>2016</td>
</tr>
<tr>
<td>SDRGO</td>
<td>2018</td>
</tr>
<tr>
<td>SDRGW</td>
<td>2019</td>
</tr>
<tr>
<td>SDROG</td>
<td>2020</td>
</tr>
<tr>
<td>SDROW</td>
<td>2021</td>
</tr>
<tr>
<td>SDRWG</td>
<td>2022</td>
</tr>
<tr>
<td>SDRWO</td>
<td>2023</td>
</tr>
<tr>
<td>SECTBC</td>
<td>2024</td>
</tr>
<tr>
<td>SECTOR</td>
<td>2026</td>
</tr>
<tr>
<td>SEPARATE</td>
<td>2027</td>
</tr>
<tr>
<td>SEPCOND</td>
<td>2028</td>
</tr>
<tr>
<td></td>
<td>2030</td>
</tr>
<tr>
<td>SEPVALS</td>
<td>2031</td>
</tr>
<tr>
<td>SFOAM</td>
<td>2034</td>
</tr>
<tr>
<td>SGAS</td>
<td>2035</td>
</tr>
<tr>
<td>SGCR</td>
<td>2036</td>
</tr>
<tr>
<td>SGCWMIS</td>
<td>2038</td>
</tr>
<tr>
<td>SGFD</td>
<td>2039</td>
</tr>
<tr>
<td>SGF3</td>
<td>2041</td>
</tr>
<tr>
<td>SGF32D</td>
<td>2043</td>
</tr>
<tr>
<td>SGFN</td>
<td>2045</td>
</tr>
<tr>
<td>SGL,ISGL</td>
<td>2047</td>
</tr>
<tr>
<td>SGLPC</td>
<td>2048</td>
</tr>
<tr>
<td>SGOF</td>
<td>2050</td>
</tr>
<tr>
<td>SGU,ISGU</td>
<td>2052</td>
</tr>
<tr>
<td>SGWCR</td>
<td>2054</td>
</tr>
<tr>
<td>Reference</td>
<td>Page</td>
</tr>
<tr>
<td>-----------</td>
<td>------</td>
</tr>
<tr>
<td>SHRATE</td>
<td>2056</td>
</tr>
<tr>
<td>SIGMA</td>
<td>2057</td>
</tr>
<tr>
<td>SIGMAGD</td>
<td>2058</td>
</tr>
<tr>
<td>Examples</td>
<td>2059</td>
</tr>
<tr>
<td>SIGMAGDV</td>
<td>2060</td>
</tr>
<tr>
<td>Examples</td>
<td>2061</td>
</tr>
<tr>
<td>SIGMATH</td>
<td>2062</td>
</tr>
<tr>
<td>SIGMAV</td>
<td>2063</td>
</tr>
<tr>
<td>SIMULATE</td>
<td>2064</td>
</tr>
<tr>
<td>SKIP</td>
<td>2065</td>
</tr>
<tr>
<td>SKIP100</td>
<td>2066</td>
</tr>
<tr>
<td>SKIP300</td>
<td>2067</td>
</tr>
<tr>
<td>SKIPREST</td>
<td>2068</td>
</tr>
<tr>
<td>SKIPSTAB</td>
<td>2070</td>
</tr>
<tr>
<td>SLAVES</td>
<td>2071</td>
</tr>
<tr>
<td>SLGOF</td>
<td>2073</td>
</tr>
<tr>
<td>SMF</td>
<td>2075</td>
</tr>
<tr>
<td>SMRYDIMS</td>
<td>2076</td>
</tr>
<tr>
<td>SMULTX-Z</td>
<td>2077</td>
</tr>
<tr>
<td>SOCRS</td>
<td>2078</td>
</tr>
<tr>
<td>SOF2</td>
<td>2079</td>
</tr>
<tr>
<td>SOF3</td>
<td>2081</td>
</tr>
<tr>
<td>SOF32D</td>
<td>2083</td>
</tr>
<tr>
<td>SOGCRI</td>
<td>2085</td>
</tr>
<tr>
<td>SOIL</td>
<td>2087</td>
</tr>
<tr>
<td>SOILR</td>
<td>2088</td>
</tr>
<tr>
<td>SOLID</td>
<td>2089</td>
</tr>
<tr>
<td>SOLIDADS</td>
<td>2090</td>
</tr>
<tr>
<td>SOLIDMMC</td>
<td>2091</td>
</tr>
<tr>
<td>SOLIDMMS</td>
<td>2092</td>
</tr>
<tr>
<td>SOLUAQA</td>
<td>2093</td>
</tr>
<tr>
<td>SOLUBIL</td>
<td>2095</td>
</tr>
<tr>
<td>SOLUBILS</td>
<td>2097</td>
</tr>
<tr>
<td>SOLUBILT</td>
<td>2099</td>
</tr>
<tr>
<td>SOLUOC2H</td>
<td>2101</td>
</tr>
<tr>
<td>SOLUH2SC</td>
<td>2103</td>
</tr>
<tr>
<td>SOLVCONC</td>
<td>2105</td>
</tr>
<tr>
<td>SOLVDIMS</td>
<td>2106</td>
</tr>
<tr>
<td>SOLVDIRS</td>
<td>2107</td>
</tr>
<tr>
<td>SOLVENT</td>
<td>2108</td>
</tr>
<tr>
<td>SOLVFRAC</td>
<td>2109</td>
</tr>
<tr>
<td>SOLVNUM</td>
<td>2110</td>
</tr>
<tr>
<td>Examples</td>
<td>2110</td>
</tr>
<tr>
<td>SOLWNUM</td>
<td>2111</td>
</tr>
<tr>
<td>SOLWTAB</td>
<td>2112</td>
</tr>
</tbody>
</table>
ECLIPSE Reference Manual

SOLWW ................................................................. 2113
SOMGAS ........................................................... 2114
SOMWAT ........................................................... 2116
SOR ................................................................. 2118
SORBFRAc ...................................................... 2120
SORBPRES ...................................................... 2121
SORMW1 ........................................................ 2122
SOROPTS ........................................................ 2123
SORWMIS ....................................................... 2124
SOWCR ........................................................... 2125
SPECGRID ......................................................... 2127
SPECHA ........................................................... 2128
SPECHB ........................................................... 2130
SPECHEAT ....................................................... 2132
SPECHG ........................................................... 2133
SPECHH ........................................................... 2135
SPECHS ........................................................... 2137
SPECHT ........................................................... 2138
SPECHW1 ........................................................ 2139
SPECHW2 ........................................................ 2141
SPECROCK ...................................................... 2143
SPOLY .............................................................. 2144
SPREF .............................................................. 2145
SSFN ............................................................... 2146
SSHIFT ............................................................ 2147

Examples ......................................................... 2147

SSHIFTS ........................................................... 2148
SSOL ............................................................... 2149
SSOLID ............................................................ 2150
START ............................................................. 2151
STCOND .......................................................... 2152
STEST .............................................................. 2153
STHERMX1 ..................................................... 2154
STOG .............................................................. 2155
STONE ............................................................ 2157
STONE1 .......................................................... 2158
STONE1EX ...................................................... 2159
STONE2 .......................................................... 2160
STONEPAR .................................................... 2161
STOPROD ......................................................... 2164
STOREAC ......................................................... 2165
STOW ............................................................. 2166
STREF ............................................................. 2168
STRESBC ........................................................ 2169

Examples ......................................................... 2171

xxxvii
<table>
<thead>
<tr>
<th>Command</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>STRESBCL</td>
<td>2172</td>
</tr>
<tr>
<td>Examples</td>
<td>2174</td>
</tr>
<tr>
<td>STVP</td>
<td>2175</td>
</tr>
<tr>
<td>STWG</td>
<td>2176</td>
</tr>
<tr>
<td>SUMTHIN</td>
<td>2177</td>
</tr>
<tr>
<td>SURF</td>
<td>2178</td>
</tr>
<tr>
<td>SURFACT</td>
<td>2179</td>
</tr>
<tr>
<td>SURFACTW</td>
<td>2180</td>
</tr>
<tr>
<td>SURFADDW</td>
<td>2181</td>
</tr>
<tr>
<td>SURFADS</td>
<td>2182</td>
</tr>
<tr>
<td>SURFCAPD</td>
<td>2183</td>
</tr>
<tr>
<td>SURFESAL</td>
<td>2184</td>
</tr>
<tr>
<td>SURFNUM</td>
<td>2185</td>
</tr>
<tr>
<td>SURFOPTS</td>
<td>2186</td>
</tr>
<tr>
<td>SURFROCK</td>
<td>2187</td>
</tr>
<tr>
<td>SURFST</td>
<td>2188</td>
</tr>
<tr>
<td>SURFSTES</td>
<td>2189</td>
</tr>
<tr>
<td>SURFVISC</td>
<td>2190</td>
</tr>
<tr>
<td>SURFWNUM</td>
<td>2191</td>
</tr>
<tr>
<td>SWAT</td>
<td>2192</td>
</tr>
<tr>
<td>SWATINIT</td>
<td>2193</td>
</tr>
<tr>
<td>SWCR</td>
<td>2195</td>
</tr>
<tr>
<td>SWF3</td>
<td>2197</td>
</tr>
<tr>
<td>SWF32D</td>
<td>2199</td>
</tr>
<tr>
<td>SWFN</td>
<td>2201</td>
</tr>
<tr>
<td>SWGCR</td>
<td>2203</td>
</tr>
<tr>
<td>SWINGFAC</td>
<td>2205</td>
</tr>
<tr>
<td>SWL,ISWL</td>
<td>2206</td>
</tr>
<tr>
<td>SWLPC</td>
<td>2208</td>
</tr>
<tr>
<td>SWOF</td>
<td>2209</td>
</tr>
<tr>
<td>SWU,ISWU</td>
<td>2211</td>
</tr>
<tr>
<td>T</td>
<td>2213</td>
</tr>
<tr>
<td>TABDIMS</td>
<td>2214</td>
</tr>
<tr>
<td>TBASIS</td>
<td>2217</td>
</tr>
<tr>
<td>TBLK</td>
<td>2218</td>
</tr>
<tr>
<td>Examples</td>
<td>2219</td>
</tr>
<tr>
<td>TBOIL</td>
<td>2220</td>
</tr>
<tr>
<td>TBOILS</td>
<td>2221</td>
</tr>
<tr>
<td>TCBDIMS</td>
<td>2222</td>
</tr>
<tr>
<td>TCRIT</td>
<td>2223</td>
</tr>
<tr>
<td>TCRITDET</td>
<td>2224</td>
</tr>
<tr>
<td>TCRITS</td>
<td>2225</td>
</tr>
<tr>
<td>TCRITSDE</td>
<td>2226</td>
</tr>
<tr>
<td>TECBDI</td>
<td>2227</td>
</tr>
<tr>
<td>TECBDIL</td>
<td>2229</td>
</tr>
<tr>
<td>Keyword</td>
<td>Page</td>
</tr>
<tr>
<td>-------</td>
<td>------</td>
</tr>
<tr>
<td>TEMP</td>
<td>2231</td>
</tr>
<tr>
<td>TEMPI</td>
<td>2232</td>
</tr>
<tr>
<td>TEMPNODE</td>
<td>2233</td>
</tr>
<tr>
<td>TEMPVD</td>
<td>2234</td>
</tr>
<tr>
<td>TEMPFD</td>
<td>2235</td>
</tr>
<tr>
<td>THANALB</td>
<td>2236</td>
</tr>
<tr>
<td>THANALH</td>
<td>2237</td>
</tr>
<tr>
<td>THANALV</td>
<td>2238</td>
</tr>
<tr>
<td>THCGAS</td>
<td>2239</td>
</tr>
<tr>
<td>THCOIL</td>
<td>2240</td>
</tr>
<tr>
<td>THCONMF</td>
<td>2241</td>
</tr>
<tr>
<td>THCONR</td>
<td>2242</td>
</tr>
<tr>
<td>THCONSF</td>
<td>2243</td>
</tr>
<tr>
<td>THCROCK</td>
<td>2244</td>
</tr>
<tr>
<td>THCSOLID</td>
<td>2245</td>
</tr>
<tr>
<td>THCROK</td>
<td>2246</td>
</tr>
<tr>
<td>THERMAL</td>
<td>2247</td>
</tr>
<tr>
<td>THERMEX1</td>
<td>2248</td>
</tr>
<tr>
<td>THERMNUM</td>
<td>2249</td>
</tr>
<tr>
<td>THFLXTF</td>
<td>2250</td>
</tr>
<tr>
<td>THPRES</td>
<td>2251</td>
</tr>
<tr>
<td>THPRESFT</td>
<td>2252</td>
</tr>
<tr>
<td>THSTT97</td>
<td>2253</td>
</tr>
<tr>
<td>THSVC</td>
<td>2254</td>
</tr>
<tr>
<td>THTABB</td>
<td>2255</td>
</tr>
<tr>
<td>THWVC</td>
<td>2256</td>
</tr>
<tr>
<td>TIGHTEN</td>
<td>2257</td>
</tr>
<tr>
<td>TIGHTENP</td>
<td>2258</td>
</tr>
<tr>
<td>TIME</td>
<td>2259</td>
</tr>
<tr>
<td>TITLE</td>
<td>2260</td>
</tr>
<tr>
<td>TLMIXPAR</td>
<td>2261</td>
</tr>
<tr>
<td>NNUM</td>
<td>2262</td>
</tr>
<tr>
<td>Examples</td>
<td>2263</td>
</tr>
<tr>
<td>TOLCRIT</td>
<td>2264</td>
</tr>
<tr>
<td>TOPS</td>
<td>2265</td>
</tr>
<tr>
<td>TOPT</td>
<td>2266</td>
</tr>
<tr>
<td>TPAMEPS</td>
<td>2267</td>
</tr>
<tr>
<td>TPAMEPSS</td>
<td>2268</td>
</tr>
<tr>
<td>TRACER</td>
<td>2269</td>
</tr>
<tr>
<td>Examples</td>
<td>2270</td>
</tr>
<tr>
<td>TRACERKM</td>
<td>2271</td>
</tr>
<tr>
<td>TRACERKP</td>
<td>2272</td>
</tr>
<tr>
<td>TRACERS</td>
<td>2273</td>
</tr>
<tr>
<td>TRACITVD</td>
<td>2274</td>
</tr>
<tr>
<td>Command</td>
<td>Page</td>
</tr>
<tr>
<td>-----------------</td>
<td>--------</td>
</tr>
<tr>
<td>VFPCHK</td>
<td>2390</td>
</tr>
<tr>
<td>VFPIDIMS</td>
<td>2391</td>
</tr>
<tr>
<td>VFPINJ</td>
<td>2392</td>
</tr>
<tr>
<td>Examples</td>
<td>2394</td>
</tr>
<tr>
<td>VFPPDIM</td>
<td>2395</td>
</tr>
<tr>
<td>VFPPROD</td>
<td>2396</td>
</tr>
<tr>
<td>Record 1: Basic data for table</td>
<td>2397</td>
</tr>
<tr>
<td>Record 2: FLO (flow rate) values</td>
<td>2400</td>
</tr>
<tr>
<td>Record 3: THP values</td>
<td>2400</td>
</tr>
<tr>
<td>Record 4: WFR values</td>
<td>2400</td>
</tr>
<tr>
<td>Record 5: GFR values</td>
<td>2400</td>
</tr>
<tr>
<td>Record 6: ALQ values</td>
<td>2400</td>
</tr>
<tr>
<td>Records 7 to NTHP<em>NWFR</em>NGFR*NALQ+6</td>
<td>2401</td>
</tr>
<tr>
<td>Examples</td>
<td>2403</td>
</tr>
<tr>
<td>VFPTABL</td>
<td>2404</td>
</tr>
<tr>
<td>VISAGE</td>
<td>2405</td>
</tr>
<tr>
<td>VISCAQA</td>
<td>2406</td>
</tr>
<tr>
<td>Examples</td>
<td>2406</td>
</tr>
<tr>
<td>VISCD</td>
<td>2407</td>
</tr>
<tr>
<td>VISCREF</td>
<td>2408</td>
</tr>
<tr>
<td>VISDATES</td>
<td>2409</td>
</tr>
<tr>
<td>VISOPTS</td>
<td>2410</td>
</tr>
<tr>
<td>VREFW</td>
<td>2412</td>
</tr>
<tr>
<td>WADVANCE</td>
<td>2413</td>
</tr>
<tr>
<td>WAGHALT</td>
<td>2414</td>
</tr>
<tr>
<td>WAGHYSTR</td>
<td>2415</td>
</tr>
<tr>
<td>WAITBAL</td>
<td>2416</td>
</tr>
<tr>
<td>WALKALIN</td>
<td>2420</td>
</tr>
<tr>
<td>WALQCALC</td>
<td>2421</td>
</tr>
<tr>
<td>WAPI</td>
<td>2422</td>
</tr>
<tr>
<td>WARN</td>
<td>2424</td>
</tr>
<tr>
<td>WARP</td>
<td>2425</td>
</tr>
<tr>
<td>WATDENT</td>
<td>2426</td>
</tr>
<tr>
<td>WATER</td>
<td>2427</td>
</tr>
<tr>
<td>WATERTAB</td>
<td>2428</td>
</tr>
<tr>
<td>VISOPTS</td>
<td>2429</td>
</tr>
<tr>
<td>VFPPROD</td>
<td>2430</td>
</tr>
<tr>
<td>VFPPDIMS</td>
<td>2432</td>
</tr>
<tr>
<td>VFPDIM</td>
<td>2433</td>
</tr>
<tr>
<td>Record 2: FLO (flow rate) values</td>
<td>2436</td>
</tr>
<tr>
<td>Record 4: WFR values</td>
<td>2438</td>
</tr>
<tr>
<td>Record 5: GFR values</td>
<td>2439</td>
</tr>
<tr>
<td>Record 6: ALQ values</td>
<td>2440</td>
</tr>
<tr>
<td>Records 7 to NTHP<em>NWFR</em>NGFR*NALQ+6</td>
<td>2441</td>
</tr>
<tr>
<td>Examples</td>
<td>2446</td>
</tr>
<tr>
<td>Command</td>
<td>Page</td>
</tr>
<tr>
<td>-----------</td>
<td>------</td>
</tr>
<tr>
<td>WELLPROD</td>
<td>2462</td>
</tr>
<tr>
<td>WCONINJE</td>
<td>2447</td>
</tr>
<tr>
<td>WCONINJH</td>
<td>2453</td>
</tr>
<tr>
<td>WCONINJP</td>
<td>2458</td>
</tr>
<tr>
<td>WCONPROD</td>
<td>2462</td>
</tr>
<tr>
<td>WCUTBACK</td>
<td>2467</td>
</tr>
<tr>
<td>WCUTBACT</td>
<td>2471</td>
</tr>
<tr>
<td>WCYCLE</td>
<td>2474</td>
</tr>
<tr>
<td>WDFAC</td>
<td>2476</td>
</tr>
<tr>
<td>WDFACCOR</td>
<td>2478</td>
</tr>
<tr>
<td>WDRILPRI</td>
<td>2480</td>
</tr>
<tr>
<td>WDRILRES</td>
<td>2482</td>
</tr>
<tr>
<td>WDRILTIM</td>
<td>2483</td>
</tr>
<tr>
<td>WECON</td>
<td>2486</td>
</tr>
<tr>
<td>WECONCMF</td>
<td>2491</td>
</tr>
<tr>
<td>WECONINJ</td>
<td>2492</td>
</tr>
<tr>
<td>WECONT</td>
<td>2494</td>
</tr>
<tr>
<td>Examples</td>
<td>2496</td>
</tr>
<tr>
<td>WEFAC</td>
<td>2497</td>
</tr>
<tr>
<td>WELCNTL</td>
<td>2499</td>
</tr>
<tr>
<td>WELDEBUG</td>
<td>2501</td>
</tr>
<tr>
<td>WELDRAW</td>
<td>2502</td>
</tr>
<tr>
<td>WELEVNT</td>
<td>2505</td>
</tr>
<tr>
<td>WELLCF</td>
<td>2506</td>
</tr>
<tr>
<td>WELLCOMP</td>
<td>2507</td>
</tr>
<tr>
<td>Examples</td>
<td>2509</td>
</tr>
<tr>
<td>WELLDIM</td>
<td>2510</td>
</tr>
<tr>
<td>WELLGR</td>
<td>2512</td>
</tr>
<tr>
<td>WELLINJE</td>
<td>2514</td>
</tr>
<tr>
<td>Examples</td>
<td>2518</td>
</tr>
<tr>
<td>WELLKBHP</td>
<td>2520</td>
</tr>
<tr>
<td>WELLIM</td>
<td>2521</td>
</tr>
<tr>
<td>WELLOPEN</td>
<td>2522</td>
</tr>
<tr>
<td>WELLOPTS</td>
<td>2523</td>
</tr>
<tr>
<td>WELLPROD</td>
<td>2524</td>
</tr>
<tr>
<td>Examples</td>
<td>2526</td>
</tr>
<tr>
<td>WELLSHUT</td>
<td>2527</td>
</tr>
<tr>
<td>WELLSMA</td>
<td>2528</td>
</tr>
<tr>
<td>WELLSPEC</td>
<td>2529</td>
</tr>
<tr>
<td>WELLSRE</td>
<td>2531</td>
</tr>
<tr>
<td>WELLSTRW</td>
<td>2532</td>
</tr>
<tr>
<td>WELLTARG</td>
<td>2533</td>
</tr>
<tr>
<td>WELLTCB</td>
<td>2534</td>
</tr>
<tr>
<td>WELLTCBC</td>
<td>2535</td>
</tr>
<tr>
<td>WELLTCBT</td>
<td>2536</td>
</tr>
<tr>
<td>Command</td>
<td>Page</td>
</tr>
<tr>
<td>-----------</td>
<td>------</td>
</tr>
<tr>
<td>WELLWAG</td>
<td>2537</td>
</tr>
<tr>
<td>Examples</td>
<td>2538</td>
</tr>
<tr>
<td>WELMOVEL</td>
<td>2540</td>
</tr>
<tr>
<td>WELOPEN</td>
<td>2541</td>
</tr>
<tr>
<td>WELOPENL</td>
<td>2543</td>
</tr>
<tr>
<td>WELPI</td>
<td>2546</td>
</tr>
<tr>
<td>WELPRI</td>
<td>2548</td>
</tr>
<tr>
<td>WELSEGS</td>
<td>2550</td>
</tr>
<tr>
<td>Examples</td>
<td>2555</td>
</tr>
<tr>
<td>WELSEMIN</td>
<td>2557</td>
</tr>
<tr>
<td>WELSECL</td>
<td>2558</td>
</tr>
<tr>
<td>Examples</td>
<td>2562</td>
</tr>
<tr>
<td>WELSECS</td>
<td>2563</td>
</tr>
<tr>
<td>Examples</td>
<td>2567</td>
</tr>
<tr>
<td>WELTARG</td>
<td>2568</td>
</tr>
<tr>
<td>WFOAM</td>
<td>2570</td>
</tr>
<tr>
<td>WFRICSEG</td>
<td>2571</td>
</tr>
<tr>
<td>WFRICSGL</td>
<td>2572</td>
</tr>
<tr>
<td>WFRICTN</td>
<td>2573</td>
</tr>
<tr>
<td>Examples</td>
<td>2574</td>
</tr>
<tr>
<td>WFRICTNL</td>
<td>2577</td>
</tr>
<tr>
<td>Each subsequent record</td>
<td>2578</td>
</tr>
<tr>
<td>Examples</td>
<td>2578</td>
</tr>
<tr>
<td>WGASPROD</td>
<td>2581</td>
</tr>
<tr>
<td>WGMEM</td>
<td>2583</td>
</tr>
<tr>
<td>WGORPEN</td>
<td>2584</td>
</tr>
<tr>
<td>WGRUPCON</td>
<td>2586</td>
</tr>
<tr>
<td>WH2NUM</td>
<td>2588</td>
</tr>
<tr>
<td>WH3NUM</td>
<td>2589</td>
</tr>
<tr>
<td>WHEDREFD</td>
<td>2590</td>
</tr>
<tr>
<td>WHISTCTL</td>
<td>2591</td>
</tr>
<tr>
<td>Examples</td>
<td>2592</td>
</tr>
<tr>
<td>WHTEMP</td>
<td>2593</td>
</tr>
<tr>
<td>WI</td>
<td>2594</td>
</tr>
<tr>
<td>WINJEDET</td>
<td>2595</td>
</tr>
<tr>
<td>WINJGAS</td>
<td>2596</td>
</tr>
<tr>
<td>WINJ MIX</td>
<td>2598</td>
</tr>
<tr>
<td>WINJMULT</td>
<td>2600</td>
</tr>
<tr>
<td>WINJOIL</td>
<td>2602</td>
</tr>
<tr>
<td>WINJORD</td>
<td>2603</td>
</tr>
<tr>
<td>WINJ TEMP</td>
<td>2605</td>
</tr>
<tr>
<td>WINJW</td>
<td>2607</td>
</tr>
<tr>
<td>WINJWAT</td>
<td>2608</td>
</tr>
<tr>
<td>WLI T</td>
<td>2609</td>
</tr>
<tr>
<td>Command</td>
<td>Page</td>
</tr>
<tr>
<td>--------------------</td>
<td>------</td>
</tr>
<tr>
<td>WLIFTOPT</td>
<td>2613</td>
</tr>
<tr>
<td>WLIMTOL</td>
<td>2616</td>
</tr>
<tr>
<td>WLIST</td>
<td>2617</td>
</tr>
<tr>
<td>WLISTARG</td>
<td>2619</td>
</tr>
<tr>
<td>WLISTDYN</td>
<td>2621</td>
</tr>
<tr>
<td>Examples</td>
<td>2623</td>
</tr>
<tr>
<td>WLISTNAM</td>
<td>2624</td>
</tr>
<tr>
<td>WMF</td>
<td>2626</td>
</tr>
<tr>
<td>WMFVDD</td>
<td>2627</td>
</tr>
<tr>
<td>WNAME</td>
<td>2628</td>
</tr>
<tr>
<td>WNETCTRL</td>
<td>2629</td>
</tr>
<tr>
<td>WNETDP</td>
<td>2631</td>
</tr>
<tr>
<td>WORKLIM</td>
<td>2632</td>
</tr>
<tr>
<td>WORKTHP</td>
<td>2633</td>
</tr>
<tr>
<td>WPAVE</td>
<td>2634</td>
</tr>
<tr>
<td>Examples</td>
<td>2635</td>
</tr>
<tr>
<td>Calculation of well block average pressures</td>
<td>2636</td>
</tr>
<tr>
<td>WPAVEDEPE</td>
<td>2638</td>
</tr>
<tr>
<td>WPIMULT</td>
<td>2639</td>
</tr>
<tr>
<td>WPIMULTL</td>
<td>2641</td>
</tr>
<tr>
<td>WPITAB</td>
<td>2643</td>
</tr>
<tr>
<td>WPLUG</td>
<td>2644</td>
</tr>
<tr>
<td>WPOLYMER</td>
<td>2646</td>
</tr>
<tr>
<td>WPOLYRED</td>
<td>2647</td>
</tr>
<tr>
<td>WPOTCALC</td>
<td>2648</td>
</tr>
<tr>
<td>WREGROUP</td>
<td>2649</td>
</tr>
<tr>
<td>WRFT</td>
<td>2651</td>
</tr>
<tr>
<td>Examples</td>
<td>2651</td>
</tr>
<tr>
<td>WRFTPLT</td>
<td>2652</td>
</tr>
<tr>
<td>Examples</td>
<td>2655</td>
</tr>
<tr>
<td>WSALT</td>
<td>2656</td>
</tr>
<tr>
<td>Examples</td>
<td>2657</td>
</tr>
<tr>
<td>WSCCLEAN</td>
<td>2658</td>
</tr>
<tr>
<td>WSCCLENLS</td>
<td>2660</td>
</tr>
<tr>
<td>WSCCTAB</td>
<td>2662</td>
</tr>
<tr>
<td>WSEGAICD</td>
<td>2664</td>
</tr>
<tr>
<td>WSEGDFIN</td>
<td>2666</td>
</tr>
<tr>
<td>WSEGDFMD</td>
<td>2670</td>
</tr>
<tr>
<td>WSEGDFPA</td>
<td>2672</td>
</tr>
<tr>
<td>WSEGDFMS</td>
<td>2675</td>
</tr>
<tr>
<td>WSEGEXSS</td>
<td>2676</td>
</tr>
<tr>
<td>WSEGFGLIM</td>
<td>2680</td>
</tr>
<tr>
<td>WSEGFMOD</td>
<td>2683</td>
</tr>
<tr>
<td>WSEGHEAT</td>
<td>2686</td>
</tr>
<tr>
<td>WSEGINIT</td>
<td>2689</td>
</tr>
<tr>
<td>Variable</td>
<td>Page</td>
</tr>
<tr>
<td>-----------</td>
<td>------</td>
</tr>
<tr>
<td>WSEGLABY</td>
<td>2693</td>
</tr>
<tr>
<td>WSEGLINK</td>
<td>2697</td>
</tr>
<tr>
<td>WSEGMULT</td>
<td>2699</td>
</tr>
<tr>
<td>WSEGPROP</td>
<td>2701</td>
</tr>
<tr>
<td>WSEGPULL</td>
<td>2703</td>
</tr>
<tr>
<td>WSEGSEP</td>
<td>2706</td>
</tr>
<tr>
<td>WSEGSICD</td>
<td>2708</td>
</tr>
<tr>
<td>WSEGSOVL</td>
<td>2716</td>
</tr>
<tr>
<td>WSEGVALV</td>
<td>2721</td>
</tr>
<tr>
<td>WSEGWELL</td>
<td>2725</td>
</tr>
<tr>
<td>WSEPCOND</td>
<td>2726</td>
</tr>
<tr>
<td>WSF</td>
<td>2727</td>
</tr>
<tr>
<td>WSOLVENT</td>
<td>2728</td>
</tr>
<tr>
<td>WSRFACT</td>
<td>2729</td>
</tr>
<tr>
<td>WTADD</td>
<td>2730</td>
</tr>
<tr>
<td>WTAKEGAS</td>
<td>2732</td>
</tr>
<tr>
<td>WTEMP</td>
<td>2734</td>
</tr>
<tr>
<td>WTENPQ</td>
<td>2735</td>
</tr>
<tr>
<td>WTEST</td>
<td>2736</td>
</tr>
<tr>
<td>WTHPMAX</td>
<td>2739</td>
</tr>
<tr>
<td>WTMULT</td>
<td>2741</td>
</tr>
<tr>
<td>WTRACER</td>
<td>2743</td>
</tr>
<tr>
<td>WVFPDP</td>
<td>2745</td>
</tr>
<tr>
<td>WVFPEXP</td>
<td>2746</td>
</tr>
<tr>
<td>Notes</td>
<td>2747</td>
</tr>
<tr>
<td>Example</td>
<td>2748</td>
</tr>
<tr>
<td>WWPAVE</td>
<td>2749</td>
</tr>
<tr>
<td>X</td>
<td>2752</td>
</tr>
<tr>
<td>XMF</td>
<td>2753</td>
</tr>
<tr>
<td>XMFVP</td>
<td>2754</td>
</tr>
<tr>
<td>XSTC</td>
<td>2755</td>
</tr>
<tr>
<td>Y</td>
<td>2756</td>
</tr>
<tr>
<td>YLDNUM</td>
<td>2757</td>
</tr>
<tr>
<td>YMF</td>
<td>2758</td>
</tr>
<tr>
<td>YMFVP</td>
<td>2759</td>
</tr>
<tr>
<td>YOUNGMOD</td>
<td>2760</td>
</tr>
<tr>
<td>YSTC</td>
<td>2761</td>
</tr>
<tr>
<td>Z</td>
<td>2762</td>
</tr>
<tr>
<td>ZCORN</td>
<td>2763</td>
</tr>
<tr>
<td>ZCRIT</td>
<td>2764</td>
</tr>
<tr>
<td>ZCRITDET</td>
<td>2765</td>
</tr>
<tr>
<td>ZCRITS</td>
<td>2766</td>
</tr>
<tr>
<td>ZCRITSDE</td>
<td>2767</td>
</tr>
</tbody>
</table>
New Developments

The 2014.1 release of the ECLIPSE suite includes some major developments enhancing the functionality, usability and performance capabilities of the simulators. We have focused on additions to chemical EOR, consistency and robustness of our relative permeability hysteresis models in addition to general client requests. The release also includes performance improvements with a new solver in both the ECLIPSE 100 and ECLIPSE 300 simulators. Licensing has also been significantly simplified, many options now being included into the standard eclipse license, allowing for more ready accessibility to simulator functions. The addition of a new parallel option specifically designed for workstation use is also offered as an alternative to the fully feature parallel/MR feature.

- The changes made to licensing in this release are described.
- The new facilities which have been added to the simulators for this release are described for ECLIPSE 100 and ECLIPSE 300.
- Behavioral changes, which may change the results of a simulation, or alter the form or content of the output, have been made for ECLIPSE 100 and ECLIPSE 300.
- The new keywords added in this release are described for ECLIPSE 100 and ECLIPSE 300.
- Some keywords have been altered in this release and these changes are described for ECLIPSE 100 and ECLIPSE 300.

Licensing changes

There have been significant changes to licensing in this release.

ECLIPSE 100 licensing changes

- An alternative license option for parallel simulation has been introduced called the "block parallel" license. This option allows access to the PARALLEL option at a level suitable for current desktop machines without having to invest in the fully flexible parallel licenses. The BPARA keyword has been added to request that a block parallel license is used rather than standard parallel licenses. A simulation can be run with up to eight parallel domains using a single block parallel license. Block parallel licenses may not be "stacked" to run more than eight parallel domains.
- From 2014.1 the following licenses are no longer required or checked out. Functionality previously covered by these licenses is now available as part of the base license.
ECLIPSE 300 licensing changes

- An alternative license option for parallel simulation has been introduced called the "block parallel" license. This option allows access to the PARALLEL option at a level suitable for current desktop machines without having to invest in the fully flexible parallel licenses. The BPARA keyword has been added to request that a block parallel license is used rather than standard parallel licenses. A simulation can be run with up to eight parallel domains using a single block parallel license. Block parallel licenses may not be "stacked" to run more than eight parallel domains.

- From 2014.1 the following licenses are no longer required or checked out. Functionality previously covered by these licenses is now available as part of the base license.
  - CO₂ Storage
  - Coal Bed Methane Model
  - Flux Boundary Conditions

New facilities

ECLIPSE 100

The new facilities in ECLIPSE 100 are described in the following sections:

- RUNSPEC section
- GRID section
- PROPS section
- REGIONS section
- SUMMARY section

RUNSPEC section

- The CPR keyword has been added to activate the CPR linear solver, which employs the Constrained Pressure Residual method to solve the linear equations. This can often be useful for more difficult problems, when the standard ORTHOMIN linear solver takes a large number of linear iterations to converge.
• The **MESSAGES** keyword now has a 13\(^{th}\) item which controls the maximum number of times some messages are output to the PRT file.

**GRID section**

• The **MINNNCT** keyword has been extended to allow the user to specify a minimum diffusivity and thermal transmissibility for a non-neighbor connection as well as (or instead of) a minimum transmissibility.

  Additionally, the logic has been improved so that values below the threshold are not preserved just because the NNC is preserved for other reasons.

• The **ROCKFRAC** keyword, previously supported in ECLIPSE 300 only, has been added to ECLIPSE 100 to simplify the input of cell volume fractions representing rock in Coal Bed Methane or Shale Gas model runs.

• Item 215 of the **OPTIONS** keyword can now be used to replace unset enumerated values with the value from an adjacent cell. This can be useful in coarsened models where a representative cell is inactive in the original model.

**PROPS section**

• The keywords **SURFSTES** and **ESSNODE** are added to activate the salinity dependent Surfactant model which allows the water-oil surface tension to vary with salinity as well as surfactant concentration. These tables can be output to the PRINT file with **RPTPROPS** argument **SURF**.

• The keyword **LANGMPL** can be used to scale the pressure used to calculate the adsorption capacities for a Coal Bed Methane Model.

• The keyword **EHYSTR** now has a 13\(^{th}\) item. If set to 1, a modification of the Killough method for the calculation of the hysteresis scanning curve on the wetting phase relative permeability is activated. This modification will, in general, ensure that the scanning curve remains within the region enclosed by the drainage and imbibition bounding curves. This correction is only applicable to the Killough wetting phase relative permeability hysteresis model, in both water-wet and oil-wet systems; if specified with other hysteresis models, it will be ignored.

• The end-point scaling keywords **LSWL**, **LSWLPC**, **LSWCR**, **LSWU**, **LSOWCR**, **LSOGCR**, **LKRW**, **LKRWR**, **LKRO**, **LKRORW**, **LKRORG** and **LPCW** have been added to permit scaling of the low salinity (oil-wet) water and oil saturation table end-points when the **LOWSALT** and **ENDSCALE** options have been specified.

• The end-point scaling keywords **HWSWL**, **HWSWLPC**, **HWSWCR**, **HWSWU**, **HWSOWCR**, **HWSOGCR**, **HWKRW**, **HWKRW**, **HWKRO**, **HWKROKRW**, **HWKROKRORG** and **HWPCW** have been added to permit scaling of the surfactant (high salinity) water-wet water and oil saturation table end-points when the **SURFACTW** and **ENDSCALE** options have been specified.

• The end-point scaling keywords **LWSWL**, **LWSWLPC**, **LWSWCR**, **LWSWU**, **LWSOWCR**, **LWSOGCR**, **LWKRW**, **LWKRW**, **LWKRO**, **LWKROKRW**, **LWKROKRORG** and **LWPCW** have been added to permit scaling of the low salinity water-wet water and oil saturation table end-points when the **LOWSALT**, **SURFACTW** and **ENDSCALE** options have been specified.

• The polymer keyword **PLYATEMP** has been added to control temperature dependence of the polymer adsorption tables (keywords **PLYADS** or **PLYADSS**) in the **PROPS** section.)
**REGIONS section**

- The saturation region keywords `LSNUM`, `HWSNUM` and `LWSNUM` have been introduced as aliases to the existing keywords `LWSLTNUM`, `SURFWNUM` and `LSTWNUM` respectively to provide a more evident association between the saturation region and the end-point scaling array keyword for models utilizing the `ENDSCALE`, `LOWSALT` and `SURFACTW` options.

**SUMMARY section**

- The SUMMARY section keyword `BTSTSUR` that returns the water-oil surface tension that depends on surfactant concentration (when `SURFST` is used) can now also be used to output values of water-oil surface tension that depends on surfactant and salt concentrations (when `SURFSTES` and `ESSNODE` are used).
- The SUMMARY section keywords `BFLOW0I, BFLOW0J, BFLOW0K, BPSHLZI, BPSHLZJ, BPSHLZK, BSHWVISI, BSHWVISJ, BSHWVISK, BVELWOI, BVELWOJ` and `BVELWOK` may now be used in conjunction with the `PLYSHLOG` option. These mnemonics were previously compatible only with the `PLYSHLOG` option.
- The SUMMARY section keywords `MLINEARP, MSUMLINP, NCPRLINP` and `NLINEARP` have been added to output performance data for the pressure solve, when using the `CPR` linear solver.
- The SUMMARY section keywords `WPIO, WPIG, WPIW` and `WPIL` have been added to output the well productivity index for the oil, gas, water and liquid phases respectively.
- The SUMMARY section keyword `CPRL` has been added to output the average connection pressure for lumped completions.
- The SUMMARY section keywords `SSFR` and `SSCN` are now also valid for the salinity dependent surfactant model.
- The SUMMARY keywords `BSRTW0I, BSRTW0J, BSRTW0K, BSRTWI, BSRTWJ` and `BSRTWK` now display the water shear rates prior to and following shear effects in the relevant direction, for models that combine the brine option with the polymer shear option activated using the `PLYSHLOG` and `SHRATE` keywords.

**SCHEDULE section**

- The keyword `CSKIN` has been added to allow connection skin factors to be updated using a user-defined argument.

**ECLIPSE 300**

The new facilities in ECLIPSE 300 are described in the following sections:

- RUNSPEC section
- GRID section
- PROPS section
- SOLUTION section
- SUMMARY section
- SCHEDULE section
RUNSPEC section

• The **WAGHALT** keyword has been provided to select an alternative WAG hysteresis model which conforms more closely with the ECLIPSE 100 WAG hysteresis model both in terms of the saturation direction change transitions and in the application of conventional hysteresis to those phases not subject to WAG hysteresis.

• The **CPR** keyword has been added to activate the CPR linear solver, which employs the Constrained Pressure Residual method to solve the linear equations. This can often be useful for more difficult problems, when the standard WARP linear solver takes a large number of linear iterations to converge.

GRID section

• The **MINNNCT** keyword has been extended to allow the user to specify a minimum diffusivity and thermal transmissibility for a non-neighbor connection as well as (or instead of) a minimum transmissibility.

  Additionally, the logic has been improved so that values below the threshold are not preserved just because the NNC is preserved for other reasons.

PROPS section

• A polymer model, similar to the ECLIPSE 100 Polymer Flood option, can now be used in ECLIPSE 300. The new model is described in the "Polymer Flood Model" in the *ECLIPSE Technical Description*. The model is activated when at least one water component is identified as a polymer using the **CWTYPE** keyword.

• Item 4 of keyword **REACOPS** has been added to allow the user to specify a scaling factor controlling the reaction rate of zero-order reactants in equilibrium deviation reactions (see **EQLDREAC** keyword).

• A salinity dependent Surfactant model can be activated using keywords **SURFSTES** and **ESSNODE** to allow the water-oil surface tension to vary with salinity as well as surfactant concentration. The **SURFSTES** tables can be output to the PRINT file with **RPTPROPS** argument **SURFSTES**.

• The effective salinity calculation model required for the salinity dependent Surfactant and Polymer models, is activated using keyword **SALTEFF**. The salinity nodes can be input with the keywords **ESSNODE** and **ESPNODE**, respectively. The **ESSNODE** and **ESPNODE** tables can be output to the PRINT file with **RPTPROPS** arguments **ESSNODE** and **ESPNODE**.

• The Alkaline model can now be activated with the Surfactant model by setting at least one water component as an alkaline with the **CWTYPE** keyword and using keyword **ALSURFST**. The **ALSURFST** table can be output to the PRINT file with **RPTPROPS** argument **ALSURFST**.

• The keyword **LANGMPL** can be used to scale the pressure used to calculate the adsorption capacities for a Coal Bed Methane or Shale Gas model.

• The keyword **EHYSTR** has now a 13th item. If set to 1, a modification of Killough method for the calculation of the hysteresis scanning curve on the wetting phase relative permeability is activated. This modification will, in general, ensure that the scanning curve remains within the region enclosed by the drainage and imbibition bounding curves. This correction is only applicable to the Killough wetting phase relative permeability hysteresis model, in both water-wet and oil-wet systems; if specified with other hysteresis models, it will be ignored.

• The keywords **PLYVISC** and **PLYVISCS** may be used to specify the polymer viscosity multiplier functions. These tables can be output to the PRINT file with **RPTPROPS** arguments **PLYVISC** and **PLYVISCS**, respectively.
The keyword PLYOPTS may be used to set optional parameters for the "Polymer Flood Model" in the ECLIPSE Technical Description.

**SOLUTION section**

- The arguments POLY and POLYVM have been added to keywords RPTSOL and RPTRST to output polymer concentration and polymer viscosity multiplier in polymer flood studies, respectively.
- The arguments ESALTP and ESALTS have been added to keywords RPTSOL and RPTRST to output effective salinity values calculated using inputs to keyword SALTEFF in polymer and surfactant flood studies, respectively.

**SUMMARY section**

- BSURFST and BSURFCP may now also be used to output water-oil surface tension and capillary pressure respectively as functions of surfactant concentration and effective salinity of the water phase when the salinity dependent Surfactant model is activated by keywords SURFSTES and ESSNODE.
- BPOLY and BPOLYVM may be used to output the block values of polymer concentration and polymer viscosity multiplier respectively when the Polymer Flood Model is active. See the "Polymer Flood Model" in the ECLIPSE Technical Description.
- BESALTP and BESALTS may be used to output the block values of effective salinity calculated as defined with keyword SALTEFF if item 1 is set to POLY or SURF, respectively.
- BALSURF and BALSTML may now be used to output alkaline concentration and alkaline surface tension multiplier determined from data provided in ALSURFST.
- The SUMMARY section keywords WPIL has been added to output the well productivity index for the liquid phase.
- The SUMMARY section keyword CPRL has been added to output the average connection pressure for lumped completions.
- The SUMMARY section keywords MLINEARP, MSUMLINP, NCPRLINS and NLINEARP have been added to output performance data for the pressure solve, when using the CPR linear solver.

**SCHEDULE section**

- The keywords PLYVISC and PLYVISCS may be used to update the polymer viscosity multipliers previously specified in the PROPS section.
- The keyword CSKIN has been added to allow connection skin factors to be updated using a user-defined argument.
- The arguments POLY and POLYVM have been added to keywords RPTSCHED and RPTRST to output polymer concentration and polymer viscosity multiplier in polymer flood studies, respectively.
- The arguments ESALTP and ESALTS have been added to keywords RPTSCHED and RPTRST to output effective salinity values calculated using inputs to keyword SALTEFF in polymer and surfactant flood studies, respectively.
Behavioral changes

**ECLIPSE 100**

The behavioral changes in ECLIPSE 100 are described in the following sections:

- **RUNSPEC section**
- **GRID section**
- **EDIT section**
- **PROPS section**
- **REGIONS section**
- **SOLUTION section**
- **SCHEDULE section**

**RUNSPEC section**

- When using the **SURFACT** and **DUALPORO** models together, a contribution to the capillary number is now made for the matrix-fracture connection. The behavior can be reverted by setting item 211 of the **OPTIONS** keyword to 1.

- When using the **SURFACT** or **LOWSALT** models with **GRAVDR**, the calculation for component cell heights now uses end-points interpolated from the relevant saturation tables. The behavior can be reverted by setting item 214 of the **OPTIONS** keyword to 1.

- The **GRIDOPTS** can now only be specified once. Pre-2014.1 behavior can be restored by setting item 223 of the **OPTIONS** keyword to 1.

- When the **PARALLEL** option is in use, messages output to the PRT files from the slave processors will be copied to the master PRT file. The pre-2014.1 behavior can be restored by setting item 219 of the **OPTIONS** keyword to 1.

- Item 13 has been added to the **MESSAGES** keyword to set the print limit for some frequently repeated output messages. This limit is set to 10 by default but can be ignored by setting item 220 of the **OPTIONS** keyword to 1.

**GRID section**

- Values set in a work array by the **OPERATE** or **OPERATER** keyword are now preserved to the end of the **GRID** section, or until a keyword is encountered which defines or switches to a different grid (such as **CARFIN**). Prior to 2014.1, values were reset to zero by each invocation of the keyword.

- The logic used to determine how a non-neighbor connection with very small transmissibility should be treated has been improved and standardized across NNC types. The new behavior is described in more detail in the **MINNNCT** keyword, which can be used to customize the minimum values.

  This may lead to slight differences in exactly how the number of non-neighbor connections is reported in the PRT file, with more connections created initially and then those which are too small for the new standard test being discarded.

  Pre-2014.1 behavior can be restored by setting item 221 of the **OPTIONS** keyword to 1.
• Field units of volumes COALV and PORVF (Coal Bed Methane option) output to the INIT file via RPTINIT arguments with the same names have been changed from FT3 to RB in order to unify their output with ECLIPSE 300 and postprocessors. For the same reason, COALV and PORVF values previously output as undefined are now reported as zero. These changes do not affect output to the PRT file (keywords RPTGRID and RPTGRIDL) which remains as in the previous releases. In addition to these changes, it is now possible to output COALV and PORVF to the INIT file for LGRs in parallel runs.

• The Flux Boundary Conditions option (keyword USEFLUX) can no longer be used in decks containing POLYMER, BRINE, SURFACTANT, ALKALINE, SOLVENT or TRACER. Pre-2014.1 behavior can be restored by setting item 232 of the OPTIONS keyword to 1.

• The corner point geometry neighbor transmissibility calculations have been enhanced to ensure that cells which do not physically touch can never have a non-zero neighbor transmissibility between them. Pre-2014.1 behavior can be restored by setting item 236 of the OPTIONS keyword to 1.

EDIT section

• Values set in a work array by the OPERATE or OPERATER keyword are now preserved to the end of the EDIT section, or until a keyword is encountered which defines or switches to a different grid (such as REFINE). Prior to 2014.1, values were reset to zero by each invocation of the keyword.

PROPS section

• Connate gas can be included in the end-point scaling calculation of the saturation at which KRWR occurs in the end-point scaled space by setting item 212 of the OPTIONS keyword to 1.

By default connate gas is omitted from these calculations.

• Corrections have been made to the calculation of water relative permeability when used as a component in the calculation of the three-phase oil relative permeability for Stone's Second Model STONE2 when end-point scaling and hysteresis have been specified. Pre-2014.1 behavior can be restored by setting item 213 of the OPTIONS keyword to 1.

• Values set in a work array by the OPERATE or OPERATER keyword are now preserved to the end of the PROPS section, or until a keyword is encountered which defines or switches to a different grid (such as REFINE). Prior to 2014.1, values were reset to zero by each invocation of the keyword.

• For 2014.1, new versions of the LOWSALT, SURFACT and SURFACTW options are available which support separate sets of end-point scaling keywords for scaling the water and oil table saturation functions associated with the high salinity (oil-wet) (SWL etc.), the low salinity (oil-wet) (LSWL etc.), the (high salinity) (water-wet) (HWSWL etc.) and the low salinity water-wet (LWSWL etc.) saturation regions. Prior to 2014.1, end-point scaling was only available for these options via the single set of end-point scaling keywords SWL etc. and this was only applied to the high salinity saturation table end-points. Pre-2014.1 behavior can be selected by setting item 225 of the OPTIONS keyword to 1.

• If the pre-2014.1 version of the LOWSALT, SURFACT and SURFACTW options is selected by setting item 225 of the OPTIONS keyword to 1, a correction has been applied to ensure that the minimum water saturations specified via the end-point scaling keyword SWL are used for the calculation of the oil and water relative permeabilities. This correction may be reverted by setting item 210 of the OPTIONS keyword to 1.
- The value of inertial coefficient specified via VDFLOW is now applied to all saturation regions when two or more regions have been specified. Pre-2014.1 behavior, for which the value of inertial coefficient was only applied to the first saturation region, can be restored by setting item 217 of the OPTIONS keyword to 1.

- In 2014.1, the diffusive flow equation expressed in FIELD units has been modified. The pre 2014.1 behavior is restored by multiplying diffusion coefficients by 5.614583, which is the conversion factor between bbl and ft³. Diffusion coefficients are input through the DIFFC keyword. METRIC and LAB unit systems remain unchanged.

- For the 2014.1 version, corrections have been made to the input processing of the two-dimensional capillary pressure data specified via the PCG32D and PCW32D keywords. Prior to 2014.1, only the first table was used for all saturation regions. If more than one table was specified, the second and subsequent tables were ignored. For the 2014.1 version, successive tables are read and allocated to their associated saturation regions. Pre-2014.1 behavior can be restored by setting item 233 of the OPTIONS keyword to 1.

- For the 2014.1 version, corrections have been made to the calculation of the saturation used to determine the gas relative permeability at the residual saturation of the displacing phase (KRGR) for gas-water models with end-point scaling. Prior to the 2014.1 version, this residual saturation was calculated using the critical oil in water saturation. For the 2014.1 version, this is calculated using the critical water saturation. Pre-2014.1 behavior can be restored by setting item 234 of the OPTIONS keyword to 1.

- For the 2014.1 version, Killough hysteresis for water-wet systems has been modified. Corrections have been made to the calculation of the wetting phase relative permeability when subject to hysteresis and end-point scaling. Pre-2014.1 behavior can be restored by setting item 222 of the OPTIONS keyword to 1.

- For the 2014.1 version, Killough hysteresis for oil-wet systems has been modified. Corrections have been made to the calculation of the two phase oil to water relative permeability, and connate water saturation has been included in the calculation of oil to gas relative permeability. Pre-2014.1 behavior can be restored by setting item 229 of the OPTIONS keyword to 1.

- For the 2014.1 version, corrections have been made in the inverse lookup (used when modeling hysteresis) with three-point end-point scaling (SCALECRS keyword activated). Pre-2014.1 behavior can be restored by setting item 231 of the OPTIONS keyword to 1.

- By default, prior to polymer shear calculations, ECLIPSE 100 now includes the mobile water saturation in the conversion from flow rate to velocity and, for runs including keyword SHRATE, applies a modified conversion factor from velocity to shear rate.
  - If item 228 of OPTIONS is set to 1, this activates the pre-2014.1 conversion from flow rate to velocity in which the mobile water saturation is excluded and may be applied to models that use either PLYSHEAR or PLYSHLOG.
  - If OPTIONS item 228 is set to 2, this activates the pre-2014.1 conversion from velocity to shear rate and may only be applied to models that use keyword SHRATE.
  - If OPTIONS item 228 is set to 3, this activates the pre-2014.1 conversions from flow rate to velocity and from velocity to shear rate and may only be applied to models that use keyword SHRATE.

Please see "Polymer Flood Model" in the ECLIPSE Technical Description for precise details of the forms of these conversions.
REGIONS section

• Values set in a work array by the OPERATE or OPERATER keyword are now preserved to the end of the REGIONS section, or until a keyword is encountered which defines or switches to a different grid (such as REFINE). Prior to 2014.1, values were reset to zero by each invocation of the keyword.

SOLUTION section

• The calculation of the threshold pressure when defaulting item 3 of the THPRES keyword will be calculated from the initial conditions considering any phase present in the upstream cell. Pre 2014.1 behavior can be restored by setting item 238 of the OPTIONS keyword to 1.

SCHEDULE section

• An error will now be emitted in parallel when a well is defined using WELSPECL on an LGR that is switched off. This can be downgraded to a warning by setting item 216 of the OPTIONS to 1.

• For the 2014.1 version, when either VDFLOW or VDFLOWR is specified, the non-Darcy component is not applied by default to the phase mobility calculations in the well connections under any circumstance. Prior to the 2014.1 version, the non-Darcy component was only applied to the well connections if the FHERCHBL keyword was specified. Pre-2014.1 behavior can be restored by setting item 218 of OPTIONS to 1. Alternatively if this item is set to 2, the non-Darcy component calculated from the VDFLOW or VDFLOWR keyword will be applied to the wells unless the WDFAC or WDFACCOR keywords or item 12 of the COMPDAT (or item 13 of the COMPDATL keyword have been specified.

• Connection ordering of RFT and PLT data (see WRFTPLT) is now controlled by the keyword COMPORD for standard wells, and always follows the TRACK method for multisegment and friction wells. Prior to 2014.1 connections were always output in the order they were input. This behavior can be reproduced setting item 2 of COMPORD to 'INPUT'.

• When UDAs are used as arguments in the keyword CECON the values are now only applied to those connections selected in the keyword. In versions earlier than 2014.1, the value is applied to all connections in the well. The pre-2014.1 behavior may be restored by setting item 224 of keyword OPTIONS to 1.

• For polymer, brine and solvent runs a minor change has been made to the 'TRNC' timestepping control. Pre-2014.1 behavior can be restored by setting item 230 of the OPTIONS keyword to 1.

• The use of the WSALT keyword in dual porosity models with local grid refinements is no longer allowed. This restriction can be overridden by setting item 237 of the OPTIONS keyword to 1; this is not recommended.

• Some changes have been made to the TRACER solution method to improve convergence in parallel runs when NNCs are present. This could affect any quantity based on passive tracers or environmental tracers. Note that this includes, but is not limited to, BRINE, when not used with POLYMER and LOWSALT.

ECLIPSE 300

The behavioral changes in ECLIPSE 300 are described in the following sections:

• RUNSPEC section

• GRID section
RUNSPEC section

- The multi-component water option activated with the COMPW keyword, cannot be used when the Flux boundary option is used (DUMPFLUX or USEFLUX keywords).

GRID section

- The logic used to determine how a non-neighbor connection with very small transmissibility should be treated has been improved and standardized across NNC types. The new behavior is described in more detail in the MINNNCT keyword, which can be used to customize the minimum values.

This may lead to slight differences in exactly how the number of non-neighbor connections is reported in the PRT file, with more connections created initially and then those which are too small for the new standard test being discarded.

Pre-2014.1 behavior can be restored by setting item 311 of the OPTIONS3 keyword to 1.

- The Flux boundary option (DUMPFLUX or USEFLUX keywords) is not compatible with the multi-component water option which is activated with the COMPW keyword.

PROPS section

- For models with end-point scaling specified, corrections have been made to the scaling algorithms for the case where the saturations at which the end-points KRWR, KRGR, KORW and KORG (relative permeabilities at residual saturation of the displacing phase) occur are the same as the saturations at which KRW, KRG, KRO and KRO (relative permeabilities at maximum saturation) respectively occur. This also applies to the imbibition counterparts if hysteresis is being modeled. Pre-2014.1 behavior can be selected by setting item 306 of the OPTIONS3 keyword to 1.

- For models which use the ODD3P coupled three-phase relative permeability and capillary pressure hysteresis option, warning messages have been suppressed where the oil-water or gas-oil primary capillary pressure curves cross the secondary (increasing) or tertiary (decreasing) capillary pressure curves. Pre-2014.1 behavior can be selected by setting item 307 of the OPTIONS3 keyword to 1.

- Water relative permeability hysteresis for oil-wet reservoirs is now available for all reservoir models for which hysteresis is available. Prior to 2013.2 this was only available for black oil models and for other models only the drainage curve was used. Pre-2013.2 behavior can be restored by setting item 310 of the OPTIONS3 keyword to 1.

- For models which use three-point endpoint scaling of the saturation functions, (that is, where item 1 of the SCALECRS keyword has been set to YES), inverse look-up of the saturation functions will also be performed using three-point endpoint scaling. (This form of inverse look-up is typically used for hysteresis calculations.) Pre-2014.1 behavior for the inverse lookup of the saturation functions can be selected by setting item 312 of the OPTIONS3 keyword to 1.

- For models which use the WAGHYSTR keyword together with end-point scaling and alternative three-phase water relative permeability saturation tables specified via the WH3NUM keyword, a correction
has been made to prevent these defaulting to the imbibition saturation tables specified via the IMBNUM
keyword. Pre-2014.1 behavior can be selected by setting item 314 of the OPTIONS3 keyword to 1.

- For models which use Killough hysteresis for oil-wet systems (item 2 of EHYSTR keyword set to 4), a
correction has been made in the construction of scanning curves for the oil relative permeability to
gas. Pre-2014.1 behavior can be selected by setting item 316 of the OPTIONS3 keyword to 1.

- For models where hysteresis has been specified via the HYSTER option of the SATOPTS keyword in
the RUNSPEC section but the EHYSTR keyword has been omitted from the PROPS section and, in
addition, none of the alternative keywords used for specifying hysteresis, for example, HYKR, HYP,
HYST, HYSTJ or HYSTK have been specified, corrections have been made to the default hysteresis
parameter selection. For the 2014.1 version, if hysteresis has been specified via the HYSTER option of
the SATOPTS keyword but the EHYSTR keyword has been omitted, the hysteresis parameters will be
assigned to the default values specified for the EHYSTR keyword and any alternative definitions
associated with alternative hysteresis keywords will be superseded. Pre-2014.1 behavior can be
selected by setting item 320 of the OPTIONS3 keyword to 1.

- For models where negative capillary pressure end-points are specified via either the PCW or the PCG
keywords (or their imbibition counterparts) or where negative capillary pressure end-points are
calculated via the JFUNC or JFUNCR keywords, these values will now be used in the simulation.
Prior to 2014.1, if negative capillary pressure end-points were defined via either of these mechanisms,
the capillary pressure end-point values used would default to the corresponding saturation table
values. Pre-2014.1 behavior can be selected by setting item 321 of the OPTIONS3 keyword to 1.

**SOLUTION section**

- When using enumeration to set the pressure, it is now an error if values are not set explicitly for all
active global cells by the end of the SOLUTION section. Pre-2014.1 behavior (where the pressure in
such cells was defaulted to atmospheric pressure) can be selected by setting item 313 of the
OPTIONS3 keyword to 1.

**SUMMARY section**

- Improved consistency of open/shut status reporting for production wells subject to economic limits
(keyword WECON).

- Region flow rates and totals for wells with cross flow now account for connections that are still net
injectors at the point where the well is opened.

- Wells retested for economic viability using the keyword WTEST now report zero flow rates when the
well remains shut.

- The SUMMARY section mnemonic WDRPR will now return zero for wells that have left the priority
drilling queue.

**SCHEDULE section**

- Connection ordering of RFT and PLT data (see WRFTPLT) is now controlled by the keyword
COMPORD for standard wells, and always follows the TRACK method for multisegment wells. Prior to
2014.1 connections were always output in the order they were input. This behavior can be reproduced
setting item 2 of COMPORD to 'INPUT'.

- For the 2014.1 version, in models with more than one equation of state region, the calculation of the
generalized pseudo pressure properties for a given well completion utilizes the equation of state
number associated with the grid block containing the completion rather than the equation of state number associated with the well.

Pre-2014.1 behavior (which utilizes the equation of state number associated with the well rather than the completion) can be restored by setting item 315 of the OPTIONS3 keyword to 1.

• For the 2014.1 version, if the effective well bore radius, set with WELSPECS item 7 and COMPDAT item 9, is less than the actual radius, the effective radius will be set to the actual radius. This is in line with the ECLIPSE 100 behavior. The pre-2014.1 behavior can be restored by setting item 317 of the OPTIONS3 keyword to 1. The ECLIPSE 100 behavior may be changed to the pre-2014.1 ECLIPSE 300 behavior by setting item 226 of the OPTIONS keyword to 1.

• For the 2014.1 version, an additional check has been added to avoid negative discriminants in the solution of the quadratic inflow rate used to calculate the D-Factor multiplier for wells or well completions for which D-Factors have been specified via the WDFAC keyword, the WDFACCOR keyword or item 12 of the COMPDAT keyword (or item 13 of the COMPDATL keyword). The pre-2014.1 behavior can be restored by setting item 319 of the OPTIONS3 keyword to 1.

• For the 2014.1 version, if the MULTX keyword is modified using MULTIREG, the new multipliers are reapplied to the transmissibility only in the region which has been changed. This also applies to all combinations of MULTX, MULTY and MULTZ when modified using MULTIREG, ADDRREG and EQUALREG.

• From 2014.1, for the multi-component water option (COMPW), the initial composition of analytic aquifers must be specified using the AQSTREAW keyword.

• From 2013.2 additional checks have been put in place to ensure that user-defined relative permeabilities for injector completions in locally refined grids defined with the COMPKRIL keyword are correctly applied. The pre-2013.2 behavior can be restored by setting item 309 of the OPTIONS3 keyword to 1.

New keywords

ECLIPSE 100

The new keywords in ECLIPSE 100 are described in the following sections:

• RUNSPEC section
• GRID section
• PROPS section
• REGIONS section
• SUMMARY section
• SCHEDULE section

RUNSPEC section

The new RUNSPEC section keywords are:

• CPR Activates the CPR linear solver.
• BPARA Requests a block parallel license.
GRID section
The new GRID section keywords are:
- **ROCKFRAC** Inputs rock volume fractions for the Coal Bed Methane or Shale Gas model.

PROPS section
The new PROPS section keywords are:
- **SURFSTES** Water-oil surface tension versus surfactant and salt concentrations
- **ESSNODE** Salt concentration nodes for water-oil surface tension
- **LANGMPL** Pressure multiplier when computing adsorption capacities for the Coal Bed Methane model.
- **LSWL, LSWLPC, LSWU, LSOWCR, LSOGCR, LKRW, LKRWR, LKRO, LKRORW, LKROG** and **LPCW** Low salinity (oil-wet) water and oil saturation table end-points.
- **HWSWL, HWSWLPC, HWSWCR, HWSWU, HWSOWCR, HWSOGCR, HWKRW, HWKWR, HWKRO, HWKROG, HWKROG** and **HWPCW** Adsorbed surfactant (high salinity) water-wet water and oil saturation table end-points.
- **LWSWL, LWSWLPC, LWSWCR, LWSWU, LWSOWCR, LWSOGCR, LWKRW, LWKWR, LWKRO, LWKROG, LWKROG** and **LWP CW** Adsorbed surfactant low salinity water-wet water and oil saturation table end-points.
- **PLYATEMP** Temperature dependence of the polymer adsorption tables.

REGIONS section
The new REGIONS section keywords are:
- **LSNUM** Low salinity (oil-wet) saturation region numbers (abbreviated alias for existing keyword **LWSLTNUM**).
- **HWSNUM** (High salinity) water-wet saturation region numbers (abbreviated alias for existing keyword **SURFWNUM**).
- **LWSNUM** Low-salinity water-wet saturation region numbers (abbreviated alias for existing keyword **LSLTWNUM**).

SUMMARY section
The new SUMMARY keywords are:
- **WPIO** Oil phase productivity index
- **WPIG** Gas phase productivity index
- **WP IW** Water phase productivity index
- **WPIL** Liquid phase productivity index
- **CPRL** Average connection pressure in a lumped completion
• MLINEARP, MSUMLINP, NCPRLINS and NLINEARP may now be used to output performance data for the pressure solve, when using the CPR linear solver.

**SCHEDULE section**

The new SCHEDULE keywords are:

• **CSKIN** Updates well connection skin factors.

**ECLIPSE 300**

The new keywords in ECLIPSE 300 are described in the following sections:

• **RUNSPEC section**

• **PROPS section**

• **SUMMARY section**

• **SCHEDULE section**

**RUNSPEC section**

The new RUNSPEC section keywords are:

• **WAGHALT** Selects alternative WAG hysteresis model.

• **CPR** Activates the CPR linear solver.

• **BPARA** Requests a block parallel license.

**PROPS section**

The new PROPS section keywords are:

• **PLYVISC** Provides a table of water phase viscosity multipliers as a function of polymer concentration.

• **PLYVISCSC** Provides a table of water phase viscosity multipliers as a function of polymer concentration for salt sensitive studies.

• **ESPNODE** Provides a table of effective salinity nodes for salt sensitive polymer flood studies.

• **PLYOPTS** Polymer model options.

• **SURFSTES** Provides a table of water-oil surface tension versus surfactant and salt concentrations.

• **ESSNODE** Salt concentration nodes for water-oil surface tension.

• **ALSURFST** Surfactant model: provides a table of water-oil surface tension multipliers as a function of alkaline concentration.

• **SALTEFF** Provides a means to compute effective salinity values for polymer and surfactant flood studies.

• **LANGMPL** Pressure multiplier when computing adsorption capacities for the Coal Bed Methane or Shale Gas model.
SUMMARY section

The new SUMMARY section keywords are:

- **WPIL** Liquid phase productivity index.
- **CPRL** Average connection pressure in a lumped completion.
- **BPOLY** Polymer concentration, measured as the mass of polymer per volume of water phase (Polymer option).
- **BPOLYVM** Polymer viscosity multiplier (Polymer option).
- **BALSURF** and **BALSTML** may now be used to output alkaline concentration and alkaline surface tension multiplier determined from data provided in **ALSURFST**.
- **BESALTS** and **BESALTP** may now be used to output effective salinity values computed using data specified by **SALTEFF** for Surfactant and Polymer models, respectively.
- **MLINEARP**, **MSUMLINP**, **NCPRLINS** and **NLINEARP** may now be used to output performance data for the pressure solve, when using the **CPR** linear solver.

SCHEDULE section

The new SCHEDULE keywords are:

- **PLYVISC** Provides a table of water phase viscosity multipliers as a function of polymer concentration.
- **PLYVISCS** Provides a table of water phase viscosity multipliers as a function of polymer concentration for salt sensitive studies.
- **CSKIN** Updates well connection skin factors.

Altered keywords

ECLIPSE 100

The altered keywords in ECLIPSE 100 are described in the following sections:

- RUNSPEC section
- GRID section
- PROPS section
- SOLUTION section
- SCHEDULE section
- OPTIONS keyword

RUNSPEC section

The altered RUNSPEC keywords in ECLIPSE 100 are:
MESSAGES

New item for setting the print limit for some frequently repeated output messages. This limit is set to 10 by default but will be ignored if item 220 of the OPTIONS keyword is set to 1.

GRID section

The altered GRID keywords in ECLIPSE 100 are:

- **RPTGRID** New argument ROCKFRAC enables reporting of rock volume fractions for the Coal Bed Methane and Shale Gas options.
- **RPTGRIDL** New argument ROCKFRAC enables reporting of rock volume fractions for the Coal Bed Methane and Shale Gas options.
- **RPTINIT** New argument ROCKFRAC enables reporting of rock volume fractions for the Coal Bed Methane and Shale Gas options.
- **MINNNCT** In addition to a minimum transmissibility, it is now possible to specify a minimum diffusivity and/or thermal transmissibility below which a non-neighbor connection will be deleted. Connections are only deleted if all three properties are below the required minimum values.

PROPS section

The altered PROPS keywords in ECLIPSE 100 are:

- **EHYSTR** Setting item 13 to 1 enables the construction of the wetting phase relative permeability Killough's hysteresis scanning curve from a restrained imbibition curve. This will prevent errant behavior of the scanning curve, in cases where the original scanning curve deviates from the region bounded by imbibition and drainage curves.
- **RPTPROPS** When existing argument SURF is requested, the output additionally includes the water-oil surface tension table (SURFSTES) and the effective salinity nodes table (ESSNODE).

SOLUTION section

The altered SOLUTION keywords in ECLIPSE 100 are:

- **RPTSOL** When existing argument SURFBLK is requested, the output additionally includes water-oil surface tension (SURFST or SURFSTES).

SCHEDULE section

The altered SCHEDULE keywords in ECLIPSE 100 are:

- **RPTSCHED** When existing argument SURFBLK is requested, the output additionally includes water-oil surface tension (SURFST or SURFSTES).
- The use of the WSALT keyword in dual porosity models with local grid refinements is no longer allowed. This restriction can be overridden by setting item 237 of the OPTIONS keyword to 1; this is not recommended.

OPTIONS keyword

The altered OPTIONS are:
<table>
<thead>
<tr>
<th>Switch</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>191</td>
<td>This item has been deprecated. To control the number of messages output for dead block connections use item 13 of the <strong>MESSAGES</strong> keyword or item 220 of the <strong>OPTIONS</strong> keyword.</td>
</tr>
<tr>
<td>210</td>
<td>If set to 1, the pre-2014.1 behavior is restored for the calculation of relative permeabilities when using the <strong>SURFACT</strong> or <strong>LOWSALT</strong> models with <strong>ENDSCALE</strong>.</td>
</tr>
<tr>
<td>211</td>
<td>If set to 1, the pre-2014.1 behavior is restored for the calculation of the capillary number when using the <strong>SURFACT</strong> and <strong>DUALPORO</strong> models.</td>
</tr>
<tr>
<td>212</td>
<td>If set to 1, connate gas will be included in the end-point scaling calculation of the saturation at which <strong>KRWR</strong> occurs in end-point scaled saturation space. By default, connate gas is omitted from these calculations.</td>
</tr>
<tr>
<td>213</td>
<td>If set to 1, the pre-2014.1 behavior is restored for the calculation of water relative permeability used as a component in the calculation of the three-phase oil relative permeability for Stone's Second Model <strong>STONE2</strong>.</td>
</tr>
<tr>
<td>214</td>
<td>If set to 1, the pre-2014.1 behavior is restored for the calculation of the component cell heights for the <strong>GRAVDR</strong> model when used with either the <strong>SURFACT</strong> or <strong>LOWSALT</strong> models.</td>
</tr>
</tbody>
</table>
| 215    | If set to 1, unset values for enumerated initial pressures in coarsened grids will be replaced with the value from an adjacent cell.  
If set to 2, unset values for all enumerated initial conditions in coarsened grids will be replaced with the value from an adjacent cell.  
This can be used to avoid issues where a pre-processor is unable to assign an enumerated value to an inactive cell which is used as the representative cell for its coarsened cell. |
<p>| 216    | If set to 1, the error emitted in parallel when a well is defined using <strong>WELSPCL</strong> on an LGR that is switched off is downgraded to a warning. |
| 217    | If set to 1, the pre-2014.1 behavior for <strong>VDFLOW</strong> is restored such that the value of the inertial coefficient for non-Darcy flow is only applied to the first saturation region when more than one saturation region has been defined. For the 2014.1 version, the inertial coefficient is applied to all saturation regions. |
| 218    | If set to 1, the pre-2014.1 behavior is restored for the non-Darcy flow component specified via the <strong>VDFLOW</strong> or <strong>VDFLOWR</strong> keyword. Prior to 2014.1 this was only applied to the phase mobility calculations for the well connections if the <strong>FHERCHBL</strong> keyword had also been specified. For the 2014.1 version, by default, the non-Darcy flow component due to the <strong>VDFLOW</strong> or <strong>VDFLOWR</strong> keyword is not applied to the well connections in any circumstance. If the <strong>VDFLOW</strong> or <strong>VDFLOWR</strong> flow component is required to be applied to the well connections, this may be enabled by setting this <strong>OPTIONS</strong> item to 2. Wells or well completions for which D-Factor calculations have been specified via the <strong>WDFAC</strong> keyword the <strong>WDFACCOR</strong> keyword or item 12 of the <strong>COMPDAT</strong> keyword (or item 13 of the <strong>COMPDATL</strong> keyword) will supersede those due to the <strong>VDFLOW</strong> or <strong>VDFLOWR</strong> keyword. |
| 219    | If set to 1, the pre-2014.1 message output format for parallel runs is used; messages from slave processors will not appear in the output files for the master process. |
| 220    | If set to 1, the message output limit from <strong>MESSAGES</strong> item 13 is ignored. |</p>
<table>
<thead>
<tr>
<th>Switch</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>221</td>
<td>If set to 1, the pre-2014.1 behavior for determining when a non-neighbor connection should be deleted based on its transmissibility is restored. Additionally, values which are below the cutoff value for all of transmissibility, diffusivity and thermal transmissibility will be preserved provided that the NNC itself is preserved. Note that, if this option is used, the diffusivity and thermal transmissibility settings from the <code>MINNNCT</code> keyword will have no effect.</td>
</tr>
<tr>
<td>222</td>
<td>If set to 1, the pre–2014.1 behavior for Killough hysteresis model for water-wet systems will be selected. For the 2014.1 version, corrections have been made to the calculation of the wetting phase relative permeability when subject to hysteresis and end-point scaling.</td>
</tr>
<tr>
<td>223</td>
<td>If set to 1, the pre 2014.1 behavior of allowing multiple uses of the <code>GRIDOPTS</code> keyword is restored.</td>
</tr>
<tr>
<td>224</td>
<td>If set to 1, the pre-2014.1 behavior for <code>CECON</code> is restored. UDAs used in <code>CECON</code> are applied to all connections in the wells specified with the particular UDA.</td>
</tr>
<tr>
<td>225</td>
<td>If set to 1, the pre 2014.1 behavior for modeling the saturation functions for low salinity <code>LOWSALT</code>, surfactant <code>SURFACT</code> and surfactant wettability <code>SURFACTW</code> effects is restored. For the 2014.1 version, extensions have been made to facilitate the end-point scaling of the low salinity saturation functions together with a number of amendments to the calculation process.</td>
</tr>
<tr>
<td>226</td>
<td>If set to 1, the effective well bore radius, used in the calculation of the well productivity index, can be larger than the drainage radius. Otherwise, the ratio of the two radii will be set to unity in the calculation of the productivity index. For the ECLIPSE 300 equivalent please refer to item 317 of the <code>OPTIONS3</code> keyword.</td>
</tr>
<tr>
<td>227</td>
<td>If set to 1, the preferred phase of an injection well is not enforced to match the actual injection phase. For the ECLIPSE 300 equivalent please refer to item 318 of the <code>OPTIONS3</code> keyword.</td>
</tr>
<tr>
<td>228</td>
<td>By default, prior to polymer shear calculations, ECLIPSE 100 now includes the mobile water saturation in the conversion from flow rate to velocity and, for runs including keyword <code>SHRATE</code>, applies a modified conversion factor from velocity to shear rate. If <code>OPTIONS</code> item 228 is set to 1, this activates the pre-2014.1 conversion from flow rate to velocity in which the mobile water saturation is excluded and may be applied to models that use either <code>PLYSHLOG</code> or <code>PLYSHLOG</code>. If <code>OPTIONS</code> item 228 is set to 2, this activates the pre-2014.1 conversion from velocity to shear rate and may only be applied to models that use keyword <code>SHRATE</code>. If <code>OPTIONS</code> item 228 is set to 3, this activates the pre-2014.1 conversions from flow rate to velocity and from velocity to shear rate and may only be applied to models that use keyword <code>SHRATE</code>. Please see the chapter entitled &quot;Polymer Flood Model&quot; in the <em>ECLIPSE Technical Description</em> for precise details of the forms of these conversions.</td>
</tr>
<tr>
<td>229</td>
<td>If set to 1, the pre-2014.1 behavior for the Killough hysteresis model for oil-wet systems will be restored. For the 2014.1 version, corrections have been made in the two phase oil to water relative permeability, and connate water saturation is included in the calculation of oil to gas relative permeability.</td>
</tr>
<tr>
<td>230</td>
<td>If set to 1, the pre-2014.1 behavior of the 'TRNC' timestepping control for <code>POLYMER/BRINE</code> and <code>SOLVENT</code> runs will be restored.</td>
</tr>
<tr>
<td>231</td>
<td>If set to 1, the pre-2014.1 behavior for inverse lookup (used when modeling hysteresis) with three-point end-point scaling will be restored. For the 2014.1 version, corrections have been made in cases where <code>SCALECRS</code> item 1 has been set to YES.</td>
</tr>
<tr>
<td>232</td>
<td>If set to 1, this removes the ban on EOR/tracer functionality with <code>DUMPLFLUX</code> and <code>USEFLUX</code> keywords. You must ensure that no information on EOR/tracer functionality is needed from outside the flux boundary.</td>
</tr>
</tbody>
</table>
### Table 1.1: Altered OPTIONS

#### ECLIPSE 300

The altered keywords in ECLIPSE 300 are described in the following sections:

- GRID section
- PROPS section
- SOLUTION section
- SCHEDULE section
- OPTIONS3 keyword

#### GRID section

The altered GRID keywords are:

- **MINNNCT** In addition to a minimum transmissibility, it is now possible to specify a minimum diffusivity and/or thermal transmissibility below which a non-neighbor connection will be deleted. Connections are only deleted if all three properties are below the required minimum values.
PROPS section

The altered PROPS keywords in ECLIPSE 300 are:

- **EHYSTR** Setting item 13 to 1 enables the construction of the wetting phase relative permeability Killough's hysteresis scanning curve from a restrained imbibition curve. This will prevent errant behavior of the scanning curve, in cases where the original scanning curve deviates from the region bounded by imbibition and drainage curves.

- **CWTYPE** The water component type POLY has been added to define a polymer for the Polymer Flood Model.

- **RPTPROPS** arguments:
  - **PLYVISC** outputs the polymer viscosity table specified by the PLYVISC keyword.
  - **PLYVISCS** outputs the polymer viscosity table specified by the PLYVISCS keyword.
  - **SURFSTES** outputs the polymer viscosity table specified by the SURFSTES keyword.
  - **ESPNODE** and **ESSNODE** output the effective salinity nodes table specified by the ESPNODE and ESSNODE keywords, respectively.

- **REACOPS** Item 4 enables you to specify a scaling factor used when calculating the reaction rate of zero-order reactants in equilibrium deviation reactions.

- **CWTYPE** The water component type ALK has been added to define an alkaline for the Alkaline Model.

SOLUTION section

The altered SOLUTION keywords are:

- **RPTSOL** New arguments POLY and POLYVM output polymer concentration and polymer viscosity multiplier in polymer flood studies, respectively.

- **RPTRST** New arguments POLY and POLYVM output polymer concentration and polymer viscosity multiplier in polymer flood studies, respectively.

- **RPTSOL** New arguments ESALTP and ESALTS output effective salinity values calculated using inputs to keyword SALTEFF in polymer and surfactant flood studies, respectively.

- **RPTRST** New arguments ESALTP and ESALTS output effective salinity values calculated using inputs to keyword SALTEFF in polymer and surfactant flood studies, respectively.

SCHEDULE section

The altered SCHEDULE keywords are:

- **RPTSCHED** New arguments POLY and POLYVM output polymer concentration and polymer viscosity multiplier in polymer flood studies, respectively.

- **RPTRST** New arguments POLY and POLYVM output polymer concentration and polymer viscosity multiplier in polymer flood studies, respectively.

- **RPTSCHED** New arguments ESALTP and ESALTS output effective salinity values calculated using inputs to keyword SALTEFF in polymer and surfactant flood studies, respectively.

- **RPTRST** New arguments ESALTP and ESALTS output effective salinity values calculated using inputs to keyword SALTEFF in polymer and surfactant flood studies, respectively.
**OPTIONS3 keyword**

The altered OPTIONS3 switches are:

<table>
<thead>
<tr>
<th>Switch</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>306</td>
<td>If set to 1, the pre-2014.1 behavior will be restored for models with end-point scaling specified, where the saturations at which the end-points $KRWR$, $KRGR$, $KROWN$ and $KROG$ (relative permeabilities at residual saturation of the displacing phase) occur are the same as the saturations at which $KRW$, $KRG$, $KRO$ and $KRO$ (relative permeabilities at maximum saturation) respectively occur. For the 2014.1 version, corrections have been made to prevent the maximum scaled relative permeabilities exceeding the maximum specified relative permeabilities under these circumstances. This also applies to the imbibition counterparts if hysteresis is being modeled.</td>
</tr>
<tr>
<td>307</td>
<td>If set to 1, warning messages will be issued for models which use the ODD3P coupled three-phase relative permeability and capillary pressure hysteresis option, where the oil-water or gas-oil primary capillary pressure curves cross the secondary (increasing) or tertiary (decreasing) capillary pressure curves.</td>
</tr>
<tr>
<td>308</td>
<td>If set to −1, then even when the DPGRID option is set and local grids with corner point geometry are specified, any unset corner depths in the fracture will not be copied from the matrix. This was the default behavior prior to 2014.1.</td>
</tr>
<tr>
<td>309</td>
<td>If set to 1, this will revert to the pre-2013.2 behavior of the COMPKRIL keyword for defining user relative permeabilities for injector completions in locally refined grids.</td>
</tr>
<tr>
<td>310</td>
<td>If set to 1, this will revert to the pre-2013.2 behavior for the calculation of water relative permeability hysteresis in oil-wet systems. Prior to the 2013.2 version, water relative permeability hysteresis in oil-wet systems was only available for black oil reservoir models. For other models only the drainage curve was used.</td>
</tr>
<tr>
<td>311</td>
<td>If set to 1, the pre-2014.1 behavior for determining when a non-neighbor connection should be deleted based on its transmissibility is restored. Additionally, values which are below the cutoff value for all of transmissibility, diffusivity and thermal transmissibility will be preserved provided that the NNC itself is preserved. Note that, if this option is used, the diffusivity and thermal transmissibility settings from the MINNCT keyword will have no effect.</td>
</tr>
<tr>
<td>312</td>
<td>If set to 1, this will revert to the pre-2014.1 behavior for the calculation of the inverse lookup with two-point end-point scaling. If defaulted, inverse lookup will be performed with three-point endpoint scaling.</td>
</tr>
<tr>
<td>313</td>
<td>If set to 1, this will revert to the pre-2014.1 behavior for the treatment of pressure by enumeration in cells where no value has been set, where such cells were assumed to have atmospheric pressure. If defaulted, it is an error to have an active global cell without an explicitly set pressure at the end of the SOLUTION section.</td>
</tr>
<tr>
<td>314</td>
<td>If set to 1, this will revert to the pre-2014.1 behavior for the modeling of water relative permeability hysteresis scanning curves for models using the WAGHYSTR keyword when end-point scaling has been specified and where the three-phase water relative permeability saturation table numbers have been specified via the WH3NUM keyword.</td>
</tr>
<tr>
<td>315</td>
<td>If set to 1, this will revert to the pre-2014.1 behavior for the modeling of generalized pseudo pressure in models with more than one equation of state region. For the 2014.1 version, the generalized pseudo pressure calculations for a given well completion utilize the equation of state properties associated with the grid block containing the completion rather than those associated with the well. Prior to 2014.1, the equation of state region number associated with the well rather than the completion was used.</td>
</tr>
<tr>
<td>Switch</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>316</td>
<td>If set to 1, this will revert to the pre-2014.1 behavior for the modeling of Killough hysteresis in oil-wet systems.</td>
</tr>
<tr>
<td>317</td>
<td>If set to 1, this will revert to the pre-2014.1 behavior for the calculation of well productivity indices. If set to 1, the effective well bore radius, used in the calculation of the well productivity index, can be larger than the drainage radius. Otherwise, the ratio of the two radii will be set to unity in the productivity index calculation. For the ECLIPSE 100 equivalent please refer to item 226 of the OPTIONS keyword.</td>
</tr>
<tr>
<td>318</td>
<td>If set to 1, the preferred phase of an injection well is updated to match the actual injection phase. For the ECLIPSE 100 equivalent please refer to item 227 of the OPTIONS keyword.</td>
</tr>
<tr>
<td>319</td>
<td>If set to 1, this will revert to the pre-2014.1 behavior for the application of D-Factors for wells or well completions for which values have been specified via the WDFAC keyword, the WDFACCOR keyword or item 12 of the COMPDAT keyword (or item 13 of the COMPDATL keyword). For the 2014.1 version, an additional check has been added to avoid negative discriminants in the solution of the quadratic in flow rate used to calculate the D-Factor multiplier. In all other respects, the calculation itself has not been changed.</td>
</tr>
<tr>
<td>320</td>
<td>If set to 1, this will revert to the pre-2014.1 behavior for the defaulting of the hysteresis control parameters when hysteresis has been specified via the HYSTER option of the SATOPTS keyword in the RUNSPEC section but the EHYSTR keyword has been omitted from the PROPS section. For versions prior to 2014.1, neither relative permeability or capillary pressure hysteresis were selected correctly if, in addition to omitting the EHYSTR keyword, none of the alternative keywords used for specifying hysteresis, for example, HYKR, HIYPC, HYST, HYSTJ or HYSTK were specified. For the 2014.1 version, if hysteresis has been specified via the HYSTER option of the SATOPTS keyword but the EHYSTR keyword has been omitted, the hysteresis parameters will be assigned to the default values specified for the EHYSTR keyword and any alternative definitions associated with alternative hysteresis keywords will be superseded.</td>
</tr>
<tr>
<td>321</td>
<td>If set to 1, this will revert to the pre-2014.1 behavior for the specification of negative capillary pressure end-points via the PCW and PCG keywords and their imbibition counterparts. Prior to 2014.1, if negative values were specified, the values actually used would default to the corresponding saturation table end-point values. For the 2014.1 version, if negative PCW or PCG values are specified, these will be used instead of the saturation table end-point values. This change of behavior also applies to capillary pressure end-points calculated via the JFUNC and JFUNCR keywords.</td>
</tr>
</tbody>
</table>

Table 1.2: Altered OPTIONS3 switches
An ECLIPSE data input file is split into sections, each of which is introduced by a keyword. A list of all section-header keywords is given below, together with a brief description of the contents of each section. A more detailed breakdown of the section contents may be found in the section overviews which follow immediately after this general overview.

After the section overviews, this manual contains a detailed description of the data for each keyword, in alphabetical keyword order. Some keywords are recognized by both ECLIPSE 100 and ECLIPSE 300, while others are valid in only one of the simulators. A flag table under each keyword heading indicates which simulator(s) the keyword may be used with, and the section(s) in which the keyword is entered. The flag table also indicates whether the keyword is specific to one of the ‘special extensions’ which are licensed separately.

In keywords that are recognized by both simulators, some data items may apply only to one simulator. These items are distinguished in this manual by margin notes, such as ECLIPSE 100 only. Furthermore, in data items requiring an option to be selected from a list of available options, some of the options may be valid in only one of the simulators; these options are similarly distinguished by a margin note. Margin notes such as ECLIPSE 300 are also employed to indicate that a paragraph of text applies to just one of the simulators.

### Data file sections

The data file sections are:

<table>
<thead>
<tr>
<th>Required?</th>
<th>Section Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required</td>
<td>RUNSPEC</td>
<td>Title, problem dimensions, switches, phases present and components for example.</td>
</tr>
<tr>
<td>Required</td>
<td>GRID</td>
<td>Specification of geometry of computational grid (location of grid block corners), and of rock properties (porosity and absolute permeability for example) in each grid block.</td>
</tr>
<tr>
<td>Optional</td>
<td>EDIT</td>
<td>Modifications to calculated pore volumes, grid block center depths and transmissibilities.</td>
</tr>
</tbody>
</table>
### Table 2.1: Data file sections

The sections must be specified in the order shown above.

It is recommended that the body of sections that are not frequently changed be held in separate files, which are included in the data using the `INCLUDE` keyword.

**Note:** The `OPTIMIZE` section header must not be specified in an included file.
Data file structure

Keywords

The keywords in the input data file (including section-header keywords) are each of up to 8 characters in length and must start in column 1. All characters up to column 8 are significant. Any characters on the same line as a keyword from column 9 onwards will be treated as a comment.

Keyword data

The data for a keyword should follow that keyword on a new line. The data may be split up by blanks or new lines, and position on the line is not significant.

Repeat counts

In the data following a keyword, asterisks may be used to signify repeat counts. A data quantity can be repeated a required number of times by preceding it with the required number and an asterisk. For example, the two following cases are equivalent:

<table>
<thead>
<tr>
<th>SATNUM</th>
<th>1 1 1 1 1 1 1 1 1 1 2 2 2 2 2 2 2 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>/</td>
<td></td>
</tr>
</tbody>
</table>

SATNUM 10*1 10*2 /

There must be no intervening blank spaces next to the asterisk on either side.

Default values

Certain items of data can be defaulted to a built-in default value. The keyword description indicates when defaults can be applied. There are two ways of setting quantities to their default values. Firstly, by ending a data record prematurely with a slash (/) the quantities remaining unspecified are set to their default values. Secondly, selected quantities positioned before the slash can be defaulted by entering n* where n is the number of consecutive quantities to be defaulted. For example, 3* causes the next three quantities in the keyword data to be given their default values. There must be no blank space between the number and the asterisk. If there is only one item at a time to be defaulted, then 1* must be entered. An asterisk by itself is not sufficient.

An example would be the following line, in which well PROD1 is given a minimum economic oil production rate of 2000, its minimum economic gas rate is defaulted to zero, its maximum water cut limit is set to 0.8, its maximum gas-oil ratio and water-gas ratio limits are defaulted to infinity, the workover procedure is set to shut the worst-offending connection, and all remaining items in the record are defaulted:

<table>
<thead>
<tr>
<th>WECON</th>
<th>PROD1 2000 1* 0.8 2* CON /</th>
</tr>
</thead>
<tbody>
<tr>
<td>/</td>
<td></td>
</tr>
</tbody>
</table>
Character strings

When character information, such as well names or mnemonics, is entered you may optionally enter the information within quotes. For example, the following two keyword entries are equivalent:

```
RPTSCHED
PRESSURE SOIL SGAS /
```

```
RPTSCHED
'PRESSURE' 'SOIL' 'SGAS' /
```

Quotes around strings are required when a name:
- contains embedded blanks;
- starts with a number;
- contains non-alphanumeric characters;
- uses the wildcard * character in well and group name roots, or well name lists.

Comments

Any lines beginning with the two characters ‘--’ are treated as comments, and are ignored by ECLIPSE. Comment lines (and blank lines also) may be inserted anywhere in the data file. Comments may also be added to the end of lines of data by beginning the comment with the two characters ‘--’, but in this case the comments must not contain any quotes.

Comments can also be included, without the two characters ‘’, on the same line after a slash (/) which is used to terminate a data record.

Global keywords

Some keywords may occur in any section of the input file. These keywords and brief descriptions of their functions are listed below.

Reading and echoing the input file

The ECHO and NOECHO keywords turn on and off the echoing of the input file to the print file. The initial default is echoing on.

The INCLUDE keyword enables data to be read from another named file. It is followed by the name of a file from which input is to be taken. Once read, the file is closed, and input resumed from the main file, starting from the keyword after the INCLUDE.

An example would be:

```
INCLUDE
CASE6G.DATA /
```

The SKIP keyword causes all subsequent keywords to be skipped (ignored) until an ENDSKIP keyword is encountered. The SKIP100 keyword skips the intervening keywords when running ECLIPSE 100. Similarly the SKIP300 keyword skips the intervening keywords when running ECLIPSE 300. This family of keywords is intended to ease the porting of data files between the two simulators.
The **COLUMNS** keyword can be used to reset the left and right margins for reading the input data file. Only data within these margins are read and processed. The default setting for the left and right margins is 1 and 132 respectively.

The **FORMFEED** keyword sets the formfeed character in the Print file. The default is to use standard FORTRAN carriage control.

### Setting message print and stop limits

ECLIPSE has a message handling system that has six levels of severity:

1. Messages
2. Comments
3. Warnings
4. Problems
5. Errors
6. Bugs

When one of these occurs, it is logged to the short form output, the Print file, the debug file, and optionally the message file (see the **MSGFILE** keyword). After a certain number of occurrences (the ‘print limit’) of a particular type of message the program will not print any more of that type. Also, after a certain number of occurrences (the ‘stop limit’) of a particular type of message the run automatically stops. The **MESSAGES** keyword allows you to modify the default print and stop limits.

It is possible to suppress the output of Warning messages by entering the **NOWARN** keyword, and subsequently reactivate them with the **WARN** keyword. Output of warnings for PVT and VFP table extrapolation may be controlled with **EXTRAPMS**.

### Controlling debug

The **DEBUG** and **DEBUG3** keywords control output to the debug file in ECLIPSE 100 and ECLIPSE 300 respectively. Each of the integers following the keyword controls output from a program option or turns on a test. This is intended mainly for program development purposes.

### Ending input files

The **END** keyword terminates the reading of data prior to the actual end of a input file. The program does not echo or process data after this. **END** may be used in an **INCLUDE** file. To end reading of an **INCLUDE** file prior to the actual end of file, and return control to the main input file, **ENDINC** may be used. **END** and **ENDINC** are generated automatically at the actual end of the relevant files, and so normally need not be used.

### List of global keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>E100 only</td>
<td><strong>COLUMNS</strong></td>
<td>Reset left and right margins for reading data input file</td>
</tr>
<tr>
<td>E100 only</td>
<td><strong>DEBUG</strong></td>
<td>Set debug output control switches</td>
</tr>
</tbody>
</table>
Restriction | Keyword | Description
---|---|---
E300 only | DEBUG3 | Set debug output control switches
 | ECHO | Switch on the echo of the data printed at the start of each run.
 | END | Terminate reading the data.
 | ENDINC | Terminate reading the Include file and return to the main data file.
 | ENDSKIP | Terminates skipping of keywords (SKIP, SKIP100, SKIP300).
E100 only | EXTRAPMS | Request warning messages for PVT and VFP table extrapolation
 | FORMFEED | Sets the formfeed character in the Print file
 | GETDATA | Imports named array data from previous Initial or Restart files. (GRID, EDIT, PROPS, REGIONS and SOLUTION sections.)
 | INCLUDE | Insert the contents of a specified file.
 | MESSAGES | Reset print and stop limits for messages of all severity types.
 | NOECHO | Switch off the echo of the data printed at the start of each run.
E100 only | NOWARN | Suppress warning messages
 | SKIP | Skips the following keywords until ENDSKIP is read.
 | SKIP100 | Instructs ECLIPSE 100 to skip the following keywords until ENDSKIP is read; it is ignored in ECLIPSE 300.
 | SKIP300 | Instructs ECLIPSE 300 to skip the following keywords until ENDSKIP is read; it is ignored in ECLIPSE 100.
 | WARN | Reactivate warning messages after suppression by NOWARN in ECLIPSE 100 only.

**ECLIPSE 300 error handling**

In the event of an error arising during keyword input, the offending line is displayed, with question mark (?) characters under the field that is causing difficulties. The error number quoted may be of two types:

- Up to and including 100.
  - These are errors detected within the ECLIPSE 300 data parser. Descriptions of these errors are given in table 2.2.
- Above 100.
  - System data errors, usually caused by internal read operations used to convert a character to its value.

<table>
<thead>
<tr>
<th>Error Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Unable to read next line from input file</td>
</tr>
<tr>
<td>2</td>
<td>Length of current line is zero</td>
</tr>
<tr>
<td>3</td>
<td>Quote misplaced in stack or data field</td>
</tr>
<tr>
<td>4</td>
<td>Slash character (/) found in the wrong place</td>
</tr>
<tr>
<td>5</td>
<td>Data field length greater than 24</td>
</tr>
<tr>
<td>6</td>
<td>End of file reached</td>
</tr>
<tr>
<td>Error Code</td>
<td>Description</td>
</tr>
<tr>
<td>------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>7</td>
<td>Invalid character found</td>
</tr>
<tr>
<td>8</td>
<td>Zero length stack found</td>
</tr>
<tr>
<td>9</td>
<td>Stack length 1 with illegal character</td>
</tr>
<tr>
<td>10</td>
<td>Incorrect stack pointers</td>
</tr>
<tr>
<td>11</td>
<td>Failed to find character in repeated data field</td>
</tr>
<tr>
<td>12</td>
<td>Multiplier for repeated data field has more than 24 characters</td>
</tr>
<tr>
<td>13</td>
<td>Error after internal read of multiplier value</td>
</tr>
<tr>
<td>14</td>
<td>Negative multiplier for repeated data field</td>
</tr>
<tr>
<td>15</td>
<td>Character found as first token in stack</td>
</tr>
<tr>
<td>16</td>
<td>Unrecognizable number in repeated data field</td>
</tr>
<tr>
<td>17</td>
<td>Single field has the wrong data type</td>
</tr>
<tr>
<td>18</td>
<td>Empty token next on stack</td>
</tr>
<tr>
<td>19</td>
<td>Unknown data type</td>
</tr>
<tr>
<td>20</td>
<td>Data read which is not of the type expected, and which cannot easily be converted</td>
</tr>
<tr>
<td>21</td>
<td>Error in internal read to convert character data to number</td>
</tr>
<tr>
<td>22</td>
<td>Repeated data has filed length of more than 24 characters</td>
</tr>
<tr>
<td>23</td>
<td>Error in internal read of repeated data field multiplier</td>
</tr>
<tr>
<td>24</td>
<td>More than one exponential character found</td>
</tr>
<tr>
<td>29</td>
<td>Previous data type of repeated filed not compatible for current repeated data field</td>
</tr>
<tr>
<td>30</td>
<td>Error in internal read of previous repeated data field</td>
</tr>
<tr>
<td>31</td>
<td>Unable to convert current value due to error in previous repeated data field</td>
</tr>
<tr>
<td>32</td>
<td>Values for margin setting not valid</td>
</tr>
<tr>
<td>33</td>
<td>Zero length string found internally</td>
</tr>
</tbody>
</table>

Table 2.2: Error codes reported by the ECLIPSE 300 data parser

Example data file

```
RUNSPEC
  
GRID  
RPTGRID 6*0 11*1 /  
-- GRID data is not often changed - so put it in an INCLUDE file
  INCLUDE  
    'GRID.SECT' / 
PROPS  
RPTPROPS 20*1 /  
  INCLUDE  
    'PROPS.SECT' / 
REGIONS  
RPTREGS 20*0 /  
  INCLUDE  
    'REGIONS.SECT' / 
```
RUNSPEC section

The RUNSPEC section is the first section of an ECLIPSE data input file. It contains the run title, start date, units, various problem dimensions (numbers of blocks, wells, tables and so forth), flags for phases or components present and option switches. It may be preceded only by comments, global keywords (see "Global keywords") and LOAD. The RUNSPEC section must always be present, unless the LOAD keyword is used to restart a run from a SAVE file that contains the RUNSPEC data.

The RUNSPEC section consists of a series of keywords, which turn on the various modeling options, or contain data (for example problem dimensions). For keywords that have associated data, the data record must be terminated by a slash (/). If a data record is terminated early with a slash, the remaining data items are set to their default values. Similarly, if a keyword is omitted all its associated data items are set to their default values. For most runs, the majority of the data items can be defaulted.

The minimum set of RUNSPEC keywords required by ECLIPSE 100 are:

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TITLE</td>
<td>Title</td>
</tr>
<tr>
<td>DIMENS</td>
<td>Number of blocks in X,Y,Z directions</td>
</tr>
<tr>
<td>OIL, WATER, GAS, VAPOIL, DISGAS</td>
<td>The active phases present, that is which of the saturations (R_s or R_v) vary</td>
</tr>
<tr>
<td>FIELD / METRIC / LAB</td>
<td>Unit convention</td>
</tr>
<tr>
<td>START</td>
<td>Start date of the simulation. In ECLIPSE 300 the START keyword is only mandatory if the DATES keyword is used.</td>
</tr>
<tr>
<td>WELLDIMS</td>
<td>Well and group dimensions</td>
</tr>
</tbody>
</table>

Table 2.3: Minimum RUNSPEC keywords for ECLIPSE 100

The keyword NOSIM sets data checking only, with no simulation.

All else can be defaulted to give:

- A Cartesian geometry, dispersed flow model.
- One set of PVT, Saturation, and Equilibration tables.
- One reporting region.
- Unformatted non-unified restart and graphics files.

The minimum set of keywords required by ECLIPSE 300 is similar, except that it is advisable to enter either COMPS or BLACKOIL according to whether the run is compositional or black oil. If neither is
present the **black-oil case will be assumed**. Oil and gas are included automatically, but the **WATER** keyword should be entered if there is a water phase. The **GASWAT** keyword requests water and gas phases only, using an equation of state to define the equilibrium between these phases. For thermal simulations the **THERMAL** keyword should be used with the **COMPS** keyword. An additional unit convention, **PVT-M**, is available.

It is important to distinguish between the number of tables/cells, for example, which means that the exact number of items must be provided in the data file, and the maximum number of wells/groups, for example, which merely gives an upper limit to the number of items that may be specified.

**Note:** **INCLUDE** files should only be used in the **RUNSPEC** section for datasets output using the SimOpt program.

**Note:** **INCLUDE** files should not be used with the **PARALLEL** keyword.

**ECLIPSE 100**

**Note:** To maintain compatibility with older data sets, the fixed record structure used by ECLIPSE 100 before 95A is still recognized. However, this allows a limited selection of facilities and it is recommended that you convert the **RUNSPEC** data to the current format. To help interpret older data sets, the original form of the **RUNSPEC** section is documented in "**RUNSPEC section: pre-95A fixed record structure**" in the **ECLIPSE Technical Description**. Please note that the **OPTIONS** keyword cannot be used with the fixed record structure.

### Alphabetic list of **RUNSPEC** keywords

#### Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACTDIMS</td>
<td>Dimensions for the ACTION facility.</td>
</tr>
<tr>
<td>ACTPARAM</td>
<td>Parameters for controlling ACTION facility functions.</td>
</tr>
<tr>
<td>ASPHALTE</td>
<td>Enable the asphaltene modeling</td>
</tr>
<tr>
<td>BLACKOIL</td>
<td>Requests black oil mode.</td>
</tr>
<tr>
<td>BRINE</td>
<td>Enable brine tracking.</td>
</tr>
<tr>
<td>CART</td>
<td>Specify Cartesian geometry.</td>
</tr>
<tr>
<td>COAL</td>
<td>Enable the Coal Bed Methane option.</td>
</tr>
<tr>
<td>CPR</td>
<td>Activate the CPR linear solver.</td>
</tr>
<tr>
<td>DIFFUSE</td>
<td>Enable molecular diffusion.</td>
</tr>
<tr>
<td>DIMENS</td>
<td>Specify the grid dimensions.</td>
</tr>
<tr>
<td>DUALPERM</td>
<td>Use the Dual Permeability option.</td>
</tr>
<tr>
<td>DUALPORO</td>
<td>Use the Dual Porosity option.</td>
</tr>
<tr>
<td>DYNRDIMS</td>
<td>Sets up dimensions for the Dynamic Region facility.</td>
</tr>
<tr>
<td>ENDSCALE</td>
<td>Use saturation table end point scaling.</td>
</tr>
<tr>
<td>FIELD</td>
<td>Use Field unit convention.</td>
</tr>
<tr>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>FLUXOPTS</td>
<td>Options for Flux Boundary models.</td>
</tr>
<tr>
<td>FMTHMD</td>
<td>Indicates that the HMD file is to be formatted.</td>
</tr>
<tr>
<td>FMTIN</td>
<td>Indicates that the input files are formatted.</td>
</tr>
<tr>
<td>FMTOUT</td>
<td>Indicates that the output files are to be formatted.</td>
</tr>
<tr>
<td>FULLIMP</td>
<td>Use the Fully Implicit Solution option.</td>
</tr>
<tr>
<td>GAS</td>
<td>The run contains gas.</td>
</tr>
<tr>
<td>GDIMS</td>
<td>Specify the dimensions for the instantaneous gradient option.</td>
</tr>
<tr>
<td>GRAVDR</td>
<td>Use gravity drainage in dual porosity runs.</td>
</tr>
<tr>
<td>GRAVDRB</td>
<td>Use the Vertically Discretized Gravity Drainage Model in dual porosity runs.</td>
</tr>
<tr>
<td>GRAVDRM</td>
<td>Use the Alternative Gravity Drainage Model in dual porosity runs.</td>
</tr>
<tr>
<td>GRIDOPTS</td>
<td>Set options to process grid data.</td>
</tr>
<tr>
<td>HMDIMS</td>
<td>Set dimensions for the Gradient option.</td>
</tr>
<tr>
<td>IMPES</td>
<td>Use the IMPES Solution option.</td>
</tr>
<tr>
<td>INSPEC</td>
<td>Request an Initial Index file.</td>
</tr>
<tr>
<td>LAB</td>
<td>Use Lab unit convention.</td>
</tr>
<tr>
<td>LGRCOPY</td>
<td>Copy global values to local cells before operations on cells within an LGR.</td>
</tr>
<tr>
<td></td>
<td>This will apply to all LGRs in the dataset.</td>
</tr>
<tr>
<td>MEMORY</td>
<td>Allocate the required memory at start of run.</td>
</tr>
<tr>
<td>METRIC</td>
<td>Use metric unit convention.</td>
</tr>
<tr>
<td>MISCIBLE</td>
<td>Enable the Miscible Gas Flood option (ECLIPSE 100)</td>
</tr>
<tr>
<td></td>
<td>Allow properties to depend on surface tension (ECLIPSE 300).</td>
</tr>
<tr>
<td>MONITOR</td>
<td>Request output for run-time monitoring.</td>
</tr>
<tr>
<td>MSGFILE</td>
<td>Message file output options.</td>
</tr>
<tr>
<td>MULTIN</td>
<td>Indicates that input files are multiple.</td>
</tr>
<tr>
<td>MULTOUT</td>
<td>Indicates that output files are multiple.</td>
</tr>
<tr>
<td>MULTOUTS</td>
<td>Indicates that Summary files are multiple.</td>
</tr>
<tr>
<td>MULTREAL</td>
<td>Activates Multi-Realization licensing.</td>
</tr>
<tr>
<td>NETWORK</td>
<td>Set dimensions for Extended Network Model.</td>
</tr>
<tr>
<td>NINEPOIN</td>
<td>Select the Nine-point option.</td>
</tr>
<tr>
<td>NMATRIX</td>
<td>Use the discretized Matrix Dual-porosity Model.</td>
</tr>
<tr>
<td>NOHYST</td>
<td>Disable Hysteresis option.</td>
</tr>
<tr>
<td>NOINSPEC</td>
<td>Do not write an Initial Index file.</td>
</tr>
<tr>
<td>NOMONITO</td>
<td>Disable the output for run-time monitoring.</td>
</tr>
<tr>
<td>NONNC</td>
<td>Disallow non-neighbor connections.</td>
</tr>
<tr>
<td>NORSSPEC</td>
<td>Do not write a Restart Index file.</td>
</tr>
<tr>
<td>NOSIM</td>
<td>Turn off simulation, but continue data checking.</td>
</tr>
<tr>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>NSTACK</td>
<td>Set linear solver stack size.</td>
</tr>
<tr>
<td>NUMRES</td>
<td>The grid contains multiple reservoirs.</td>
</tr>
<tr>
<td>NUPCOL</td>
<td>Number of timestep iterations in which well targets are updated.</td>
</tr>
<tr>
<td>OIL</td>
<td>The run contains oil.</td>
</tr>
<tr>
<td>PARALLEL</td>
<td>Select a parallel run</td>
</tr>
<tr>
<td>PETOPTS</td>
<td>Sets options for simulator runs fired from Petrel.</td>
</tr>
<tr>
<td>RADIAL</td>
<td>Specify radial geometry.</td>
</tr>
<tr>
<td>REGDIMS</td>
<td>Set dimensions for regions data.</td>
</tr>
<tr>
<td>ROCKCOMP</td>
<td>Enable the rock compaction option.</td>
</tr>
<tr>
<td>RSSPEC</td>
<td>Request a Restart Index file.</td>
</tr>
<tr>
<td>SATOPTS</td>
<td>Set options for directional and hysteretic relative permeabilities.</td>
</tr>
<tr>
<td>SOLID</td>
<td>Indicates a solid phase in the run</td>
</tr>
<tr>
<td>START</td>
<td>Specify the start date.</td>
</tr>
<tr>
<td>TABDIMS</td>
<td>Set table dimensions.</td>
</tr>
<tr>
<td>TITLE</td>
<td>Specify the run title.</td>
</tr>
<tr>
<td>UDADIMS</td>
<td>Dimensions for the user defined arguments facility.</td>
</tr>
<tr>
<td>UDQDIMS</td>
<td>Dimensions for the user defined quantity facility.</td>
</tr>
<tr>
<td>UDQPARAM</td>
<td>Parameters for the user defined quantity facility.</td>
</tr>
<tr>
<td>UNCODHMD</td>
<td>Enables unencoded sensitivities from the Gradient option.</td>
</tr>
<tr>
<td>UNIFIN</td>
<td>Indicates that input files are unified.</td>
</tr>
<tr>
<td>UNIFOUT</td>
<td>Indicates that output files are unified.</td>
</tr>
<tr>
<td>UNIFOUTS</td>
<td>Indicates that Summary files are unified.</td>
</tr>
<tr>
<td>VFPIDIMS</td>
<td>Set dimensions for injection well VFP tables.</td>
</tr>
<tr>
<td>VFPPDIMS</td>
<td>Set dimensions for production well VFP tables.</td>
</tr>
<tr>
<td>VISAGE</td>
<td>Activate the VISAGE geomechanics workflow.</td>
</tr>
<tr>
<td>WARP</td>
<td>Enable WARP linear solver.</td>
</tr>
<tr>
<td>WATER</td>
<td>The run contains water.</td>
</tr>
<tr>
<td>WCOMPRES</td>
<td>Reduce the memory requirements of a simulation that contains a large number of wells, some of which have many connections, but most have relatively few connections.</td>
</tr>
<tr>
<td>WELLDIMS</td>
<td>Set dimensions for well data.</td>
</tr>
<tr>
<td>WSEGDIMS</td>
<td>Set dimensions for multisegment wells.</td>
</tr>
<tr>
<td>WSEGSOLV</td>
<td>Sets parameters for the multisegment well iterative linear solver.</td>
</tr>
</tbody>
</table>
## ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>API</td>
<td>Enable the API Tracking option.</td>
</tr>
<tr>
<td>ALKALINE</td>
<td>Enable the Alkaline model.</td>
</tr>
<tr>
<td>AUTOREF</td>
<td>Set options for auto refinement.</td>
</tr>
<tr>
<td>BIGMODEL</td>
<td>Allows large models to run by changing the internal memory accounting</td>
</tr>
<tr>
<td>DISGAS</td>
<td>Run contains dissolved gas in live oil.</td>
</tr>
<tr>
<td>DISPDIMS</td>
<td>Dimensions dispersion tables</td>
</tr>
<tr>
<td>ECLMC</td>
<td>Enable the Multi-Component option.</td>
</tr>
<tr>
<td>EQLDIMS</td>
<td>Set dimensions of equilibration tables.</td>
</tr>
<tr>
<td>EQLOPTS</td>
<td>Set options for equilibration.</td>
</tr>
<tr>
<td>FAULTDIM</td>
<td>Set dimensions for fault data</td>
</tr>
<tr>
<td>FOAM</td>
<td>Enable the Foam option.</td>
</tr>
<tr>
<td>FRICTION</td>
<td>Enable the Wellbore Friction option.</td>
</tr>
<tr>
<td>GASFIELD</td>
<td>Enable special options in the Gas Field Operations Model.</td>
</tr>
<tr>
<td>GIMODEL</td>
<td>Enable the GI Pseudo-compositional option</td>
</tr>
<tr>
<td>LGR</td>
<td>Set options for local grid refinement and coarsening.</td>
</tr>
<tr>
<td>LICENSES</td>
<td>Reserve ECLIPSE 100 option licenses.</td>
</tr>
<tr>
<td>LOWSALT</td>
<td>Enable the Low-Salinity option.</td>
</tr>
<tr>
<td>NNEWTF</td>
<td>Enable the non-Newtonian modeling for polymer</td>
</tr>
<tr>
<td>NOCASC</td>
<td>Use the linear solver tracer algorithm to solve single phase tracers.</td>
</tr>
<tr>
<td>NRSOUT</td>
<td>Sets the maximum number of quantities that can be output to a restart file</td>
</tr>
<tr>
<td>OPTIONS</td>
<td>Activate special options within particular facilities in ECLIPSE 100.</td>
</tr>
<tr>
<td>PARTTRAC</td>
<td>Activate partitioned tracers for Environmental Tracers or the Surfactant option</td>
</tr>
<tr>
<td>PATHS</td>
<td>Set pathname aliases</td>
</tr>
<tr>
<td>PEDIMS</td>
<td>Used to set dimensions for a Petro-elastic Model.</td>
</tr>
<tr>
<td>PIMTDIMS</td>
<td>Set PI scaling table dimensions</td>
</tr>
<tr>
<td>POLYMER</td>
<td>Enable the Polymer Flood Model.</td>
</tr>
<tr>
<td>PSTEADY</td>
<td>Automatically runs the simulator to a steady state.</td>
</tr>
<tr>
<td>RIVRDIMS</td>
<td>Set river dimensions.</td>
</tr>
<tr>
<td>RPTHMD</td>
<td>Specify controls on the output to the HMD file.</td>
</tr>
<tr>
<td>RPTRUNSP</td>
<td>Specify controls on the output of RUNSPEC data.</td>
</tr>
<tr>
<td>SAMG</td>
<td>Enable algebraic multigrid linear solver. Note that in ECLIPSE 100 its use is restricted to single phase tracers and requires a Water Services license.</td>
</tr>
<tr>
<td>SAVE</td>
<td>Request the output of a Save file for fast restarts</td>
</tr>
<tr>
<td>SCDPDIMS</td>
<td>Set dimensions for scale deposition tables.</td>
</tr>
</tbody>
</table>
### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SMRYDIMS</td>
<td>Set maximum number of Summary file quantities.</td>
</tr>
<tr>
<td>SOLVENT</td>
<td>Enable the Four-component Solvent Model.</td>
</tr>
<tr>
<td>SURFACT</td>
<td>Enable the Surfactant Model.</td>
</tr>
<tr>
<td>SURFACTW</td>
<td>Enable modeling of changes of wettability in the Surfactant Model.</td>
</tr>
<tr>
<td>TRACERS</td>
<td>Set dimensions and options for tracers.</td>
</tr>
<tr>
<td>TEMP</td>
<td>Enable the Temperature option.</td>
</tr>
<tr>
<td>VAPOIL</td>
<td>The run contains vaporized oil in wet gas.</td>
</tr>
<tr>
<td>VE</td>
<td>Enable the Vertical Equilibrium option.</td>
</tr>
<tr>
<td>VISCD</td>
<td>Enable the Viscous Displacement option.</td>
</tr>
<tr>
<td>WPOTCALC</td>
<td>Control calculation of shut and stopped well potentials.</td>
</tr>
</tbody>
</table>

### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AIM</td>
<td>Use the AIM Solution option.</td>
</tr>
<tr>
<td>AJGRADNT</td>
<td>Use the adjoint gradient screening option.</td>
</tr>
<tr>
<td>AQUUDIMS</td>
<td>Dimensions for aquifers.</td>
</tr>
<tr>
<td>CBMOPTS</td>
<td>Specify the type of adsorption model to be used with the Coal Bed Methane option.</td>
</tr>
<tr>
<td>CO2SOL</td>
<td>Allow CO(_2) to dissolve in the aqueous phase.</td>
</tr>
<tr>
<td>CO2STORE</td>
<td>Activates the CO(_2) storage option.</td>
</tr>
<tr>
<td>COMPS</td>
<td>Requests compositional mode.</td>
</tr>
<tr>
<td>COMPSOL</td>
<td>For SOLID, BLACKOIL runs, this keyword must be used to set the number of components.</td>
</tr>
<tr>
<td>COMPW</td>
<td>Requests water component model.</td>
</tr>
<tr>
<td>DEADOIL</td>
<td>Use the Dead Oil option in thermal simulations.</td>
</tr>
<tr>
<td>DOMAIN</td>
<td>Set non-default domain size for phase property evaluation.</td>
</tr>
<tr>
<td>DPCDT</td>
<td>Set maximum rate of change of capillary pressure.</td>
</tr>
<tr>
<td>E100NOSI</td>
<td>Switch to NOSIM mode if an incompatible ECLIPSE 100 keyword is read.</td>
</tr>
<tr>
<td>EOS</td>
<td>Specify which equation of state is to be used.</td>
</tr>
<tr>
<td>EOSS</td>
<td>Specify the surface equation of state.</td>
</tr>
<tr>
<td>FASTTRAC</td>
<td>Enables shortcuts in tracer calculations.</td>
</tr>
<tr>
<td>FMTSAVE</td>
<td>Indicates that the Save files are formatted</td>
</tr>
<tr>
<td>FORMOPTS</td>
<td>Formulation options.</td>
</tr>
<tr>
<td>FREEZEPF</td>
<td>Freeze all capillary pressures in the run.</td>
</tr>
<tr>
<td>FWELLS</td>
<td>Enable the Fractured Well Completion option.</td>
</tr>
<tr>
<td>ECLIPSE 300 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>--------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td>GASSOL</td>
<td>Use the GASSOL option.</td>
</tr>
<tr>
<td>GASWAT</td>
<td>Use the water-gas model.</td>
</tr>
<tr>
<td>GEOMECH</td>
<td>Selects the geomechanics option</td>
</tr>
<tr>
<td>GEODIMS</td>
<td>Sets the number of geomechanics regions in the simulation, sets table dimensions, in terms of the number of rows and columns, for the PERMSTAB and BIOTCTAB keywords.</td>
</tr>
<tr>
<td>GPTDIMS</td>
<td>Set gas plant table dimensions.</td>
</tr>
<tr>
<td>H2SSOL</td>
<td>Allow H2S to dissolve in the aqueous phase</td>
</tr>
<tr>
<td>HEATDIMS</td>
<td>Set heater keyword dimensions in thermal simulations.</td>
</tr>
<tr>
<td>HWELLS</td>
<td>Enable the Horizontal Well Completion option.</td>
</tr>
<tr>
<td>HYKR</td>
<td>Enable the Relative Permeability Hysteresis option.</td>
</tr>
<tr>
<td>HYPIC</td>
<td>Enable the Capillary Pressure Hysteresis option.</td>
</tr>
<tr>
<td>HYST</td>
<td>Enable the Hysteresis option.</td>
</tr>
<tr>
<td>HYSTJ</td>
<td>Enable the Hysteresis option using the Jargon method.</td>
</tr>
<tr>
<td>HYSTK</td>
<td>Enable the Hysteresis option using the Killough method.</td>
</tr>
<tr>
<td>IMPSAT</td>
<td>Use the IMPSAT Solution option.</td>
</tr>
<tr>
<td>ISGAS</td>
<td>The run is gas condensate.</td>
</tr>
<tr>
<td>JALS</td>
<td>Activate the JALS linear solver.</td>
</tr>
<tr>
<td>KVALUES</td>
<td>Use K-values in the simulation</td>
</tr>
<tr>
<td>LAGTRAC</td>
<td>Activates the Lagrangian Tracer Solver option.</td>
</tr>
<tr>
<td>LIVEOIL</td>
<td>Use the live oil option in thermal simulations.</td>
</tr>
<tr>
<td>LUMPDIMS</td>
<td>Turns on, and dimensions, the LUMPING option.</td>
</tr>
<tr>
<td>MULTSAVE</td>
<td>Indicates that Save files are multiple.</td>
</tr>
<tr>
<td>MULTIPHA</td>
<td>Allows multi-phase flash for asphaltene prediction.</td>
</tr>
<tr>
<td>MEMSAVE</td>
<td>Allow use of Up Front Memory option.</td>
</tr>
<tr>
<td>NINEXZ</td>
<td>Select the Nine-point option in the XZ plane.</td>
</tr>
<tr>
<td>NINEYXZ</td>
<td>Select the Nine-point option in the YZ plane.</td>
</tr>
<tr>
<td>NOAIMLGR</td>
<td>Suppress making all refinement cells implicit in AIM runs.</td>
</tr>
<tr>
<td>NODPCO</td>
<td>No dual porosity condensation</td>
</tr>
<tr>
<td>NODPCDT</td>
<td>Disable the DPCDT option.</td>
</tr>
<tr>
<td>NODPPM</td>
<td>No dual porosity permeability multiplier.</td>
</tr>
<tr>
<td>NOFREEZE</td>
<td>Suppress use of the FREEZEPC option.</td>
</tr>
<tr>
<td>NOHYKIR</td>
<td>Do not apply hysteresis to relative permeabilities.</td>
</tr>
<tr>
<td>NOHYPC</td>
<td>Do not apply hysteresis to capillary pressure.</td>
</tr>
<tr>
<td>NOMIX</td>
<td>Do not interpolate between oil and gas relative permeabilities near the critical point.</td>
</tr>
</tbody>
</table>
### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NPROCX</td>
<td>Set number of processors in the x-direction for a parallel run.</td>
</tr>
<tr>
<td>NPROCY</td>
<td>Set number of processors in the y-direction for a parallel run.</td>
</tr>
<tr>
<td>OPTIONS3</td>
<td>Activate special options within particular facilities in ECLIPSE 300.</td>
</tr>
<tr>
<td>PDIVX</td>
<td>Select a division of the x-direction for parallel runs.</td>
</tr>
<tr>
<td>PDIVY</td>
<td>Select a division of the y-direction for parallel runs.</td>
</tr>
<tr>
<td>RESOPT</td>
<td>Activates the Reservoir Optimization license key.</td>
</tr>
<tr>
<td>PSPLITX</td>
<td>Specify multiple domain decomposition in the X-direction.</td>
</tr>
<tr>
<td>PSPLITY</td>
<td>Specify multiple domain decomposition in the Y-direction.</td>
</tr>
<tr>
<td>PSSTA</td>
<td>Use saturation pressure to assign states, instead of Li correlation.</td>
</tr>
<tr>
<td>PVT-M</td>
<td>Use PVT-metric unit convention.</td>
</tr>
<tr>
<td>REACTION</td>
<td>Define the number of chemical reactions.</td>
</tr>
<tr>
<td>ROCKDIMS</td>
<td>Set dimensions for rock over and underburden data in thermal simulations.</td>
</tr>
<tr>
<td>RSM</td>
<td>Output extra data to the RSM file.</td>
</tr>
<tr>
<td>SCFDIMS</td>
<td>Set dimensions for conductive fractures.</td>
</tr>
<tr>
<td>SECTOR</td>
<td>Define a sector model.</td>
</tr>
<tr>
<td>SECTBC</td>
<td>Specify boundary conditions for sector model.</td>
</tr>
<tr>
<td>SKIPSTAB</td>
<td>Speeds up flash calculations by skipping some one-phase stability calculations</td>
</tr>
<tr>
<td>SOLVDIMS</td>
<td>Specifies the solver dimensions for an unstructured grid.</td>
</tr>
<tr>
<td>TRACK</td>
<td>Enable the Tracking option.</td>
</tr>
<tr>
<td>TCBDIMS</td>
<td>Set temperature cutback table dimensions in thermal simulations.</td>
</tr>
<tr>
<td>THERMAL</td>
<td>Enable the Thermal option.</td>
</tr>
<tr>
<td>TRCOEF</td>
<td>Enable transport coefficients to be used.</td>
</tr>
<tr>
<td>TRPLPORO</td>
<td>Grid for triple porosity.</td>
</tr>
<tr>
<td>UNIFSAVE</td>
<td>Indicates that Save files are unified.</td>
</tr>
<tr>
<td>VECTABLE</td>
<td>Set the size of the vector property table.</td>
</tr>
<tr>
<td>VELDEP</td>
<td>Settings for velocity dependent relative permeability.</td>
</tr>
<tr>
<td>WAGHALT</td>
<td>Selects the alternative WAG hysteresis model.</td>
</tr>
</tbody>
</table>

### GRID section

The GRID section determines the basic geometry of the simulation grid and various rock properties (porosity, absolute permeability, net-to-gross ratios) in each grid cell. From this information, the program calculates the grid block pore volumes, mid-point depths and inter-block transmissibilities.

The actual keywords used depend upon the use of the radial or Cartesian geometry options. The program accepts the radial form in a Cartesian run and vice versa, but issues a warning.

The expressions used to calculate quantities related to grid geometry and rock properties are described in the *ECLIPSE Technical Description*. 
Specifying the basic grid dimensions

The reservoir geometry may be set using keywords **CART** or **RADIAL** in the **RUNSPEC** section to either

- Cartesian (X,Y,Z)
- Radial (R, Theta, Z)

and may be specified in either of two ways:

**Block Centered Geometry**

Where the blocks are horizontal and all eight corners are right angles. Each block is defined by the dimensions of its three sides and the depth of the top surface.

**Corner Point Geometry**

Where the locations of all the eight corners are provided independently and there is no requirement that all the angles of the block are right angles.

The program recognizes that the keywords **COORD** and **ZCORN** specify Corner Point Geometry. Any other specification of the grid block sizes results in Block Centered Geometry.

For complete flexibility, the **GDFILE** keyword may be used to specify all the corner point positions of each cell independently.

In addition, Petrel software can export various types of grid geometry (straight pillars, curved pillars or listric pillars) using the Generic Simulation Grid (GSG) file which can be imported using keyword **PETGRID**.

Since corner point geometry data is usually very voluminous, and difficult to enter by hand, it is recommended that a pre-processor (for example Petrel) is used to construct it. When data is prepared in this way, a **SPECGRID** keyword is included, which is read and checked, to confirm that the dimensions used by the pre-processor and simulator agree.

**Note:** Block centered and corner point geometry may not be mixed.

The number of grid blocks in each direction in the model (NX, NY, NZ) is set by the keyword **DIMENS** in the **RUNSPEC** section.

**Note:** The X and R directions have index I, the Y and Theta directions have index J, and the Z direction has index K.

All depths and thicknesses are measured along the Z axis, which is taken to be vertical, with larger values indicating greater depths.

The origin in Cartesian geometry is the top left back corner. Coordinates on the X axis are taken to increase from left to right, and on the Y axis from back to front.

In radial geometry, the origin is the center of the model, where usually a well is located. The inner radius of the reservoir must be provided. Refer to keywords **INRAD** and **RADFIN** for more information. R increases towards the right for Theta = 0; Theta increases in the clockwise direction.

The input is such that data specified using **GRID** and **EDIT** section keywords is entered always in natural order, the I-index changing most quickly, then the J-index, and the K-index most slowly. This enables the data to be read from the top left corner of a page along a line (I), then next line (J), then turn the page for the next layer (K).
The shape, size, bulk volume and position of each block is determined by block centered or corner point geometry. All other properties of the grid are the average values for the block irrespective of its geometry. Each grid block must have a porosity and a permeability in all directions where there is more than one grid block.

Data may be input as arrays with a value for each of the \( \text{NX} \times \text{NY} \times \text{NZ} \) grid blocks, or for all blocks in a part of the grid defined by the keyword \texttt{BOX}. A box exists until a new \texttt{BOX} is defined, or the keyword \texttt{ENDBOX} is read. The default \texttt{BOX} is the whole model. The areal dimensions \( X, Y, R, \) Theta may be input as vectors such as \texttt{DXV(I)} where \( DX \) varies only with \( I \) (see, for example, keyword \texttt{DXV}).

Alternatively, data may be set to constant values using the \texttt{EQUALS} keyword, and manipulated with keywords \texttt{ADD}, \texttt{MULTIPLY}, \texttt{COPY}, \texttt{MINVALUE} and \texttt{MAXVALUE}. Data can also be edited by flux region, \texttt{MULTNUM} region or \texttt{OPERNUM} region with the keywords \texttt{ADDREG}, \texttt{EQUALREG}, \texttt{COPYREG} and \texttt{MULTIREG}. The flux regions are defined with keyword \texttt{FLUXNUM}.

In block centered geometry, the dimensions of every grid block are specified using the keywords in table 2.4. In radial geometry, the outer radius may be specified (using the \texttt{OUTRAD} keyword in ECLIPSE 100, or the \texttt{RADFIN} keyword in ECLIPSE 300) instead of specifying \( DR \), and the program automatically generates spaced grid blocks in the radial direction, geometrically (for ECLIPSE 100), or logarithmically (for ECLIPSE 300). The depth of the top of each grid block is specified using the \texttt{TOPS} keyword.

### Essential keywords

<table>
<thead>
<tr>
<th>Block centered — Cartesian</th>
<th>Block centered - Radial</th>
<th>Corner point</th>
</tr>
</thead>
<tbody>
<tr>
<td>INRAD</td>
<td></td>
<td></td>
</tr>
<tr>
<td>\texttt{DXV or DX}</td>
<td>\texttt{DRV or DR or OUTRAD}</td>
<td>\texttt{COORD and ZCORN}</td>
</tr>
<tr>
<td>\texttt{DYV or DY}</td>
<td>\texttt{DTHETAV or DTHETA}</td>
<td>\texttt{or GDFILE or}</td>
</tr>
<tr>
<td>\texttt{DZ}</td>
<td>\texttt{DZ}</td>
<td>\texttt{PETGRID}</td>
</tr>
<tr>
<td>\texttt{TOPS}</td>
<td>\texttt{TOPS}</td>
<td></td>
</tr>
<tr>
<td>\texttt{PORO}</td>
<td>\texttt{PORO}</td>
<td></td>
</tr>
<tr>
<td>\texttt{PERMX}</td>
<td>\texttt{PERMR}</td>
<td>\texttt{PERMX/PERMR}</td>
</tr>
<tr>
<td>\texttt{PERMY}</td>
<td>\texttt{PERMTHT}</td>
<td>\texttt{PERMY/PERMTHT}</td>
</tr>
<tr>
<td>\texttt{PERMZ}</td>
<td>\texttt{PERMZ}</td>
<td>\texttt{PERMZ}</td>
</tr>
</tbody>
</table>

*Table 2.4: Essential GRID section keywords used to specify basic grid dimensions*

An example for a simple two layer grid 10 by 10 by 2 grid, with 200 ft. by 200 ft. by 10 ft. cells in the top layer and 200 ft. by 200 ft. cells in the bottom layer would be:

```
DX 200*200 /
DY 200*200 /
DZ 100*10 100*50 /
```

or, equivalently:

```
DXV 10*200 /
DYV 10*200 /
```
If corner point geometry is not being used, the depth as well as the dimensions of each cell must be entered. This may be done using TOPS (to specify the cell top depths) or MIDS (ECLIPSE 300 only - to specify the cell mid-point depths). In either case only the cells in the top layer need be specified; the depth of cells below may be calculated from the thickness of the cell and that of the cell above.

**Note:** MIDS should not be confused with DEPTH, used to override cell center depths after calculation in the GRID section, and described in the EDIT section overview.

In the radial case one further item of geometry information is required, namely, the inner grid block radius. This can be specified using with the INRAD keyword, or for ECLIPSE 300 only, the RADFIN keyword.

**Net-to-gross ratios**

The NTG keyword may be used to enter net-to-gross ratios for each cell. Alternatively, DZNET may be used to enter net ratios, and the program will calculate net-to-gross thicknesses as $NTG = \frac{DZNET}{DZ}$ for each cell.

**Cell porosities and pore volumes**

Cell porosities must be entered in all runs, and are used to calculate cell pore volumes. The default porosity is zero. Cells with zero pore volume are deemed inactive, and do not occupy storage during the simulation section. Porosities are entered with the PORO keyword. Note that porosities are entered as fractions rather than percentages.

Any method that results in a zero pore volume renders a cell inactive. Optionally, the ACTNUM keyword may be used to enter a 0 or 1 for each block, a 0 making the cell inactive. The criterion for an inactive cell may be reset using MINFORV (or MINPV), to set the minimum pore volume allowed:

- **MINDZNET** may be used to specify a minimum cell thickness allowed for an active cell. (ECLIPSE 300)

**Rock permeability values**

These are entered for the dimensions present using PERMX, PERMY and PERMZ (or PERMR, PERMTHT and PERM2 for radial runs).

As for other grid keywords, this could be done using the keyword explicitly, or by using the EQUALS keyword.

For example, using the grid defined above, with a permeability of 50mD in the first layer, and 33mD in the second layer, it is possible to enter:

```
PERMX 100*50 100*33 /
```

or to use the EQUALS keyword:

```
EQUALS
PERMX 50 4* 1 1 /
PERMX 33 4* 2 2 /
```
The **ADD** keyword is similar, but adds the value specified to the grid array. **MULTIPLY** multiplies the array by a given factor. **COPY** can be used to set values for one keyword equal to those for another.

A common example is setting the Y-direction permeabilities equal to the X-direction permeabilities:

```
COPY
  PERMX PERMY /
```

The **ADD**, **COPY**, **EQUALS** and **MULTIPLY** keywords are processed as they are read, so that repeat operations are possible on the same section of an array.

### Transmissibility multipliers

Once the cell dimensions and permeabilities are known, the program is in a position to calculate the transmissibilities. The transmissibilities in the program are defined so that \( T_x(I) \) is between blocks \( I \) and \( (I + 1) \). The expressions for these are given in the "Transmissibility Calculations" in the **ECLIPSE Technical Description**.

Transmissibilities may be modified by the transmissibility multiplier keywords **MULTX**, **MULTY**, **MULTZ**, **MULTR** and **MULTTHT**.

Note that the X-direction multiplier acts on the transmissibility between the cell and its neighbor in the **positive** X-direction. Similar considerations apply to the Y and Z directions.

Transmissibilities across faults identified with the **FAULTS** keyword can be modified with the keyword **MULTFLT**. Refer also to the **MULTX-**, **MULTY-**, **MULTZ-**, **MULTR-** and **MULTTHT-** keywords.

The following applies to **ECLIPSE 300**. In some **IMPES** runs, convergence can be improved by limiting the transmissibility in the Z-direction. This may be done using the **MAXTRANZ** keyword.

### Diffusivity multipliers

If the **DIFFUSE** option has been used in the **RUNSPEC** section to enable molecular diffusion to be modeled, diffusivities are calculated that reflect the effect of grid block properties upon diffusive flow. The calculation of these is described in "Diffusion" in the **ECLIPSE Technical Description**.

It is possible to set multipliers to act in these calculations, analogous to **MULTX** and similar keywords, in the normal convective flow case. The required keywords are **DIFFMX**, **DIFFMY** and **DIFFMZ** (or, for radial runs, **DIFFMR**, **DIFFMTH** and **DIFFMZ**).

As in the case of transmissibility multipliers, the diffusivity multiplier acts between the cell and its neighbor in the **positive** direction. Refer also to **DIFFMX-**, **DIFFMY-** and **DIFFMZ-** (or, for radial runs, **DIFFMR-**, **DIFFMTH-** and **DIFFMZ-**) keywords.

### BOX and ENDBOX keywords

By default, keywords that supply data over the grid (such as **DX** and **DY**) expect data for the whole grid, that is \( N_x \cdot N_y \cdot N_z \) values.
The **BOX** keyword may be used to modify this range so that values may be entered for a sub-box of cells in the reservoir. **ENDBOX** may be used to revert to the whole grid. The current box is the default range when **EQUALS**, **MULTIPLY**, **ADD** and **COPY** are used.

As an example, consider a 40 by 20 by 8 cell study, in which the first 4 layers have a porosity of 0.07 and the remaining 4 layers have a porosity of 0.03. This might be done using the **BOX** and **ENDBOX** keywords.

```plaintext
BOX
  1 40 1 20 1 4 /
  PORO
  3200*0.07 /
  ENDBOX

BOX
  1 40 1 20 5 8 /
  PORO
  3200*0.03 /
  ENDBOX
```

The **FLUXNUM** keyword may be used (even when neither **DUMPFLUX** nor **USEFLUX** is entered) to identify regions of the field by index number. Values in these regions may then be manipulated with **ADDREG**, **COPYREG**, **EQUALREG** and **MULTIREG**. The **MULTNUM** keyword and the **OPERNUM** keyword can also be used with **ADDREG**, **COPYREG**, **EQUALREG** and **MULTIREG**.

## Non-neighbor connections (NNCs)

If corner point geometry is being used, NNCs are generated to represent faults when layers overlap between columns.

There are several keywords specifically for the generation of non-neighbor flows.

### Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>COARSEN</strong></td>
<td>Generates non-neighbor connections generated from using the grid coarsening option.</td>
</tr>
<tr>
<td><strong>CIRCLE</strong></td>
<td>may be used in radial studies with more than two cells in the direction to represent flow across the 360 degree to 0 degree boundary.</td>
</tr>
<tr>
<td><strong>NOCIRCLE</strong></td>
<td>may be used in local radial grid refinement runs to complete the circle.</td>
</tr>
<tr>
<td><strong>NNC</strong></td>
<td>may be used to enter a non-neighbor transmissibility between any two cells in the reservoir. In ECLIPSE 300 this is further enhanced with the <strong>NNCGEN</strong> keyword.</td>
</tr>
<tr>
<td><strong>SIGMA</strong></td>
<td>may be used to specify a coupling parameter for NNCs set up in Dual Porosity runs. The use of this is discussed in &quot;Dual Porosity&quot; in the <em>ECLIPSE Technical Description</em>.</td>
</tr>
<tr>
<td><strong>SIGMAGD</strong></td>
<td>may be used to specify a different coupling parameter for gravity drainage. By default, the <strong>SIGMA</strong> values are used.</td>
</tr>
<tr>
<td><strong>PINCH</strong></td>
<td>specifies how missing zero thickness layers are to be treated. NNCs may be generated across the absent layer.</td>
</tr>
<tr>
<td><strong>AQUCON</strong></td>
<td>connects a numerical aquifer to the reservoir using NNCs</td>
</tr>
</tbody>
</table>

*ECLIPSE 300* By default, non-neighbor connections are not generated across pinched-out layers (or columns), unless keyword **PINCH**(or **PINCHXY**) is entered. The keywords **MINPV** and **MINPVV** may be used to specify a threshold pore volume for a cell to be inactive. Any cells whose pore volumes lie below the threshold are
made inactive. This may be used to inactivate cells with small pore volumes and to create a pinched-out layer, for example. For further details on transmissibilities see "Transmissibility Calculations" in the ECLIPSE Technical Description.

**ECLIPSE 100 only keywords**

* AQUNNC specified numerical aquifer NNC properties explicitly.

**ECLIPSE 100**

If the NEWTRAN keyword is specified, ECLIPSE 100 uses the corner point data to compute the transmissibilities across faults, and automatically generates non-neighbor connections where necessary (unless NONNC is specified in the RUNSPEC section). By default, non-neighbor connections are not generated across pinched-out layers (or columns), unless keyword PINCH, or PINCHREG (or PINCHXY) is entered.

**ECLIPSE 100**

Grids that consist of two or more unconnected regions (coupled only through group control of the wells) can be solved more efficiently as separate units, if they are identified with the ISOLNUM keyword. Refer to "Independent Reservoir Regions" in the ECLIPSE Technical Description.

**Output control**

Output to the print file of GRID section data, both input and calculated, is controlled by the RPTGRID keyword. This is followed by mnemonics for the required arrays. For example,

```
RPTGRID
  DX DY PERMX PERMY PORO /
```

requests output of X and Y cell dimensions and permeabilities, and of the porosities. The maximum and minimum values in each of the GRID arrays are automatically be printed.

**ECLIPSE 100**

Grid-valued output values in certain types of cell have the decimal point replaced with a special character for ease of identification. See 3.31 for details.

As an alternative, the ECLIPSE 100 keyword can be used. This has a series of integer values, with a 0 requesting no output of the appropriate property, an 1 (or greater) requesting output.

Output is normally generated to a GRID file, which may be read by GRAF to obtain plots of cell positions. This may be prevented, if not required, by the NOGGF keyword.

An initial file may be requested using the INIT keyword. This contains arrays such as PORV and DEPTH, which may then be read by GRAF.

The RPTINIT keyword has been added to allow the user to control GRID and EDIT output, both input and calculated, to the .INIT or .FINIT file. This is followed by mnemonics for the required arrays. For example,

```
RPTINIT
  DZ PERMX TRANY PORO /
```

requests output of Z cell dimensions, X cell permeabilities, Y cell transmissibilities and the porosities.

The output applies for both the global grid and for any LGRs contained in the model (for example Cartesian, radial, nested). The PORV information will always be output to the init file. When this keyword is specified along with a series of additional mnemonics, this information will also be output to the .INIT or .FINIT file.
Local grid refinement

Areas of the grid may be refined using the CARFIN and RADFIN keywords (for Cartesian and radial refinements, respectively).

Cells of Cartesian refinements may be further refined. The simulator treats global and refined cells together in the solution steps. In the AIM mode, local grids are made implicit unless otherwise requested.

Description of GRID section keywords

All keywords must start in column 1. All characters up to column 8 are significant.

A list of all keywords which may occur in the GRID section is given below, together with a brief description of their function. A more detailed description may be found by looking up the keyword in the manual.

Block centered geometry

- TOPS Depths of top faces of grid blocks for the current input box.
- DZ Z-direction (vertical) grid block sizes for the current input box.

Cartesian block centered

<table>
<thead>
<tr>
<th>Cartesian keywords</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DX</td>
<td>X-direction grid block sizes for the current input box (or use DXV).</td>
</tr>
<tr>
<td>DXV</td>
<td>Vector of X-direction grid block sizes or use DX. Simplified version of DX.</td>
</tr>
<tr>
<td>DY</td>
<td>Y-direction grid block sizes for the current input box (or use DYV).</td>
</tr>
<tr>
<td>DYV</td>
<td>Vector of Y-direction grid block sizes or use DY. Simplified version of DY.</td>
</tr>
</tbody>
</table>

Radial block centered

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Radial key word</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required for radial block centered geometry.</td>
<td>INRAD</td>
<td>Sets inner radius of radial grid.</td>
</tr>
<tr>
<td>Must be set</td>
<td>DR</td>
<td>R-direction grid block sizes for the current input box (or use DRV or OUTRAD).</td>
</tr>
<tr>
<td>Optional for radial geometry</td>
<td>DRV</td>
<td>Vector of R-direction grid block sizes (or use DR or OUTRAD). Simplified version of DR.</td>
</tr>
<tr>
<td>E100 only.</td>
<td>OUTRAD</td>
<td>Sets outer radius of radial grid (or use DR or DRV).</td>
</tr>
<tr>
<td>Must be set</td>
<td>DTHETA</td>
<td>Theta-direction grid block sizes for the current input box (or use DTHETAV).</td>
</tr>
<tr>
<td></td>
<td>DTHETAV</td>
<td>Vector of Theta-direction grid block sizes (or use DTHETA). Simplified version of DTHETA.</td>
</tr>
</tbody>
</table>
Corner point geometry

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Corner point keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Either COORD and ZCORN or GDFILE are required for corner point geometry.</td>
<td>COORD</td>
<td>Defines the lines that contain all grid block corner points for each (I, J) and for each reservoir in the grid.</td>
</tr>
<tr>
<td>Either COORD and ZCORN or GDFILE are required for corner point geometry.</td>
<td>ZCORN</td>
<td>Point depths of grid block corners.</td>
</tr>
<tr>
<td>Either COORD and ZCORN or GDFILE are required for corner point geometry.</td>
<td>GDFILE</td>
<td>Inputs corner point positions for each cell from a GRID file as an alternative to COORD and ZCORN.</td>
</tr>
<tr>
<td>Optional</td>
<td>ADDZCORN</td>
<td>Adds a constant value to cell corner point depths</td>
</tr>
<tr>
<td>Optional</td>
<td>EQLZCORN</td>
<td>Resets part of the corner point depth array within a specified box</td>
</tr>
<tr>
<td>E100 only</td>
<td>COORDSYS</td>
<td>Information about the coordinate system for each reservoir in the grid. Specifies completion of the circle in both corner point and block-centered geometries.</td>
</tr>
<tr>
<td>Optional</td>
<td>RESVNUM</td>
<td>Initiates COORD data input for a given reservoir in the grid</td>
</tr>
<tr>
<td>Optional</td>
<td>MAPAXES</td>
<td>Inputs map origin from a grid pre-processor, for output to the GRID file</td>
</tr>
<tr>
<td>Optional</td>
<td>MAPUNITS</td>
<td>Specifies the units used for MAPAXES data for output to the GRID file</td>
</tr>
<tr>
<td>Optional</td>
<td>GRIDUNIT</td>
<td>Specifies the grid data units that can be output by grid pre-processors or by ECLIPSE simulators writing the grid geometry files</td>
</tr>
</tbody>
</table>

Block properties, all geometries

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>PORO</td>
<td>Grid block porosities for the current input box. Must be set in some way</td>
</tr>
<tr>
<td>Optional</td>
<td>DZNET</td>
<td>Grid block net thicknesses (or use NTG).</td>
</tr>
<tr>
<td>Optional</td>
<td>NTG</td>
<td>Grid block net-to-gross ratios for current input box (or use DZNET).</td>
</tr>
<tr>
<td>Optional</td>
<td>ACTNUM</td>
<td>Identifies active grid blocks.</td>
</tr>
<tr>
<td>E300 only</td>
<td>MIDS</td>
<td>Specifies cell mid-point depths</td>
</tr>
<tr>
<td>Optional</td>
<td>MINPV</td>
<td>Minimum pore volume that each grid block must have to be active (scalar)</td>
</tr>
<tr>
<td>Optional</td>
<td>MINPVV</td>
<td>Minimum pore volumes that grid blocks in the current BOX must have to be active (vector).</td>
</tr>
</tbody>
</table>
### Transmissibility keywords

All keywords are optional.

#### Common keywords

- **NNC** Explicit entry of non-neighbor connections.
- **PINCH** Generates connections across pinched-out layers.
- **PINCHOUT** Generates connections across pinched-out layers.
- **PINCHXY** Generates connections across columns that are pinched-out horizontally.

#### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>NEWTRAN</strong></td>
<td>Specifies that corner point data should be used to compute transmissibilities.</td>
</tr>
<tr>
<td><strong>OLDTRAN</strong></td>
<td>Specifies that block center data should be used to compute transmissibilities.</td>
</tr>
<tr>
<td><strong>OLDTRANR</strong></td>
<td>Alternative calculation of transmissibilities using block characteristics.</td>
</tr>
<tr>
<td><strong>PINCHNUM</strong></td>
<td>Identifies pinchout regions for use by <strong>PINCHREG</strong>.</td>
</tr>
<tr>
<td><strong>PINCHREG</strong></td>
<td>Specifies pinchout data within <strong>PINCHNUM</strong> regions.</td>
</tr>
</tbody>
</table>
Transmissibility modifiers

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optional if radial with NR &gt; 1</td>
<td>MULTR</td>
<td>R-direction transmissibility multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if radial with NR &gt; 1</td>
<td>MULTR−</td>
<td>Negative R-direction transmissibility multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if radial with NTHETA &gt; 1</td>
<td>MULTTH</td>
<td>Theta direction transmissibility multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if radial with NTHETA &gt; 1</td>
<td>MULTTH−</td>
<td>Negative Theta-direction transmissibility multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if Cartesian with NX &gt; 1</td>
<td>MULTX</td>
<td>X-direction transmissibility multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if Cartesian with NX &gt; 1</td>
<td>MULTX−</td>
<td>Negative X-direction transmissibility multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if Cartesian with NY &gt; 1</td>
<td>MULTY</td>
<td>Y-direction transmissibility multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if Cartesian with NY &gt; 1</td>
<td>MULTY−</td>
<td>Negative Y-direction transmissibility multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if NZ &gt; 1</td>
<td>MULTZ</td>
<td>Z-direction transmissibility multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if NZ &gt; 1</td>
<td>MULTZ−</td>
<td>Negative Z-direction transmissibility multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional</td>
<td>MULTFREGT</td>
<td>Multiplies transmissibilities between regions defined with FLUXNUM or MULTNUM.</td>
</tr>
<tr>
<td>Optional</td>
<td>MULTNUM</td>
<td>Identifies regions for use with MULTFREGT, MULTREGD (ECLIPSE 100 only) and MULTREGH (ECLIPSE 100 only). Also defines regions, for editing data with ADDREG, EQUALREG, MULTIREG and COPYREG.</td>
</tr>
</tbody>
</table>

Table 2.5: Transmissibility modifiers

Faults

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FAULTS</td>
<td>Identifies fault trajectories for subsequent use by the MULTFLT keyword.</td>
</tr>
</tbody>
</table>
### Numerical aquifers

#### Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MULTFLT</td>
<td>Multipliers for the transmissibility and diffusivity across faults defined by the FAULTS keyword.</td>
</tr>
<tr>
<td>THPRESFT</td>
<td>Threshold pressures associated with faults defined by the FAULTS keyword.</td>
</tr>
</tbody>
</table>

#### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required for analytic aquifer</td>
<td>AQUANCON</td>
<td>Defines connection data for analytic aquifer</td>
</tr>
<tr>
<td>definition</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

#### Dual porosity

#### Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BTOBALFA</td>
<td>Switch on Z-direction block to block matrix-fracture coupling and supply single-value transmissibility connection fraction multiplier.</td>
</tr>
<tr>
<td>BTOBALFV</td>
<td>Switch on Z-direction block to block matrix-fracture coupling and supply multiple-value transmissibility connection fraction multipliers.</td>
</tr>
<tr>
<td>DIFFMFF</td>
<td>Diffusivity multipliers for matrix-fracture coupling.</td>
</tr>
<tr>
<td>DPGRID</td>
<td>Enables user to enter grid data for matrix cells only.</td>
</tr>
<tr>
<td>DPNUM</td>
<td>Identifies the extent of the regions in which the single porosity model may be applied in a dual porosity run.</td>
</tr>
<tr>
<td>DZMTRX</td>
<td>The vertical dimension of a typical block of matrix material, for the entire grid.</td>
</tr>
<tr>
<td>DZMTRXV</td>
<td>The vertical dimension of a typical block of matrix material, for the current box.</td>
</tr>
<tr>
<td>NMATOPTS</td>
<td>Controls the Discretized Matrix Dual Porosity option.</td>
</tr>
<tr>
<td>NODPPM</td>
<td>Specifies that permeabilities in the fracture cells are not to be multiplied by the fracture porosity.</td>
</tr>
<tr>
<td>SIGMA</td>
<td>Matrix-to-fracture coupling factor for the entire grid.</td>
</tr>
<tr>
<td>SIGMAV</td>
<td>Matrix-to-fracture coupling factor for the current box.</td>
</tr>
</tbody>
</table>
### Keyword Description

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SIGMAGD</td>
<td>Matrix-to-fracture coupling for oil-gas gravity drainage, for the entire grid</td>
</tr>
<tr>
<td>SIGMAGDV</td>
<td>Matrix-to-fracture coupling for oil-gas gravity drainage, for the current box.</td>
</tr>
</tbody>
</table>

### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LTOSIGMA</td>
<td>Provides an easy way to calculate SIGMA from LX, LY, LZ.</td>
</tr>
<tr>
<td>LX, LY, LZ</td>
<td>Inputs the representative matrix block sizes for the viscous displacement option.</td>
</tr>
</tbody>
</table>

### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MULTMF</td>
<td>Transmissibility multipliers for matrix-fracture coupling.</td>
</tr>
<tr>
<td>PERMMF</td>
<td>Permeability for matrix-fracture coupling.</td>
</tr>
</tbody>
</table>

### Compressed vertical equilibrium (ECLIPSE 100)

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optional, CVE runs only</td>
<td>COLLAPSE</td>
<td>Specifies which cells should be collapsed.</td>
</tr>
<tr>
<td>Optional, CVE runs only</td>
<td>CRITPERM</td>
<td>Sets a critical permeability to determine which cells are collapsed.</td>
</tr>
<tr>
<td>Optional, VE runs only</td>
<td>VEDEBUG</td>
<td>Controls debug output for VE options.</td>
</tr>
</tbody>
</table>

### Diffusivity modifiers

### Common keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optional if radial with &gt; 1</td>
<td>DIFFMR</td>
<td>R-direction diffusivity multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if radial with &gt; 1</td>
<td>DIFFMR-</td>
<td>Negative R-direction diffusivity multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if radial with &gt; 1</td>
<td>DIFFMTH</td>
<td>Theta-direction diffusivity multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if radial with &gt; 1</td>
<td>DIFFMTH-</td>
<td>Negative Theta-direction diffusivity multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if Cartesian with &gt; 1</td>
<td>DIFFMX</td>
<td>X-direction diffusivity multipliers for the current input box.</td>
</tr>
<tr>
<td>Restriction</td>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>--------------------------------</td>
<td>--------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Optional if Cartesian with &gt; 1</td>
<td><strong>DIFFMX</strong></td>
<td>Negative X-direction diffusivity multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if Cartesian with &gt; 1</td>
<td><strong>DIFFMY</strong></td>
<td>Y-direction diffusivity multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if Cartesian with &gt; 1</td>
<td><strong>DIFFMY</strong></td>
<td>Negative Y-direction diffusivity multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if &gt; 1</td>
<td><strong>DIFFMZ</strong></td>
<td>Z-direction diffusivity multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional if &gt; 1</td>
<td><strong>DIFFMZ</strong></td>
<td>Negative Z-direction diffusivity multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional, dual porosity runs</td>
<td><strong>DIFFMMF</strong></td>
<td>Matrix-fracture diffusivity multipliers for the current input box.</td>
</tr>
<tr>
<td>Optional</td>
<td><strong>MULTFLT</strong></td>
<td>Multipliers for the transmissibility and diffusivity across faults defined by the keyword.</td>
</tr>
</tbody>
</table>

**ECLIPSE 100 only keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optional.</td>
<td><strong>MULTREGD</strong></td>
<td>Multipliers for the diffusivity between regions defined with <strong>FLUXNUM</strong> or <strong>MULTNUM</strong>.</td>
</tr>
<tr>
<td>Optional.</td>
<td><strong>MULTNUM</strong></td>
<td>Identifies regions for use with <strong>MULTREGT</strong>, <strong>MULTREGD</strong> and <strong>MULTREGH</strong></td>
</tr>
</tbody>
</table>

**Independent reservoir regions (ECLIPSE 100)**
- **ISOLNUM** Defines the independent reservoir regions, for efficient solution.
- **RPTISOL** Generates data for the **ISOLNUM** keyword, output to the Debug file.

**Temperature option (ECLIPSE 100)**

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>MULTREGH</strong></td>
<td>Multipliers for the thermal conductivity between regions defined with <strong>FLUXNUM</strong> or <strong>MULTNUM</strong>.</td>
</tr>
<tr>
<td><strong>MULTNUM</strong></td>
<td>Identifies regions for use with <strong>MULTREGT</strong>, <strong>MULTREGD</strong> and <strong>MULTREGH</strong>.</td>
</tr>
<tr>
<td><strong>THCONR</strong></td>
<td>Rock thermal conductivities for the current input box.</td>
</tr>
</tbody>
</table>
## Nine-point scheme (ECLIPSE 300)

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NINENUM</td>
<td>Specifies the extent of the nine-point discretization scheme.</td>
</tr>
<tr>
<td>MINPORV9</td>
<td>Set a minimum pore volume for calculation of nine-point transmissibilities.</td>
</tr>
<tr>
<td>ORTHERRO</td>
<td>Set a minimum orthogonality error for calculation of nine-point transmissibilities.</td>
</tr>
<tr>
<td>ORTHERRX</td>
<td>Set a minimum orthogonality error for calculation of nine-point transmissibilities for interfaces in the X-direction.</td>
</tr>
<tr>
<td>ORTHERRY</td>
<td>Set a minimum orthogonality error for calculation of nine-point transmissibilities for interfaces in the Y-direction.</td>
</tr>
<tr>
<td>ORTHERRZ</td>
<td>Set a minimum orthogonality error for calculation of nine-point transmissibilities for interfaces in the Z-direction.</td>
</tr>
</tbody>
</table>

## Operational keywords

### Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADD</td>
<td>Adds specified constants to specified arrays in the current input box.</td>
</tr>
<tr>
<td>ADDREG</td>
<td>Adds specified constants to specified arrays in a given flux, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>BOUNDARY</td>
<td>Defines the section of the grid for which output maps are required.</td>
</tr>
<tr>
<td>BOX</td>
<td>Redefines the current input box.</td>
</tr>
<tr>
<td>COPY</td>
<td>Copies data in the current box from one specified array to another.</td>
</tr>
<tr>
<td>COPYBOX</td>
<td>Copies data from one box of cells to another box of the same size.</td>
</tr>
<tr>
<td>COPYREG</td>
<td>Copy by FLUXNUM region, MULTNUM region or OPERNUM region.</td>
</tr>
<tr>
<td>DUMPFLUX</td>
<td>Output a Flux file.</td>
</tr>
<tr>
<td>ENDBOX</td>
<td>Redefines the current input box to encompass the whole grid.</td>
</tr>
<tr>
<td>EQUALS</td>
<td>Sets specified arrays to specified constants within the current box.</td>
</tr>
<tr>
<td>EQUALREG</td>
<td>Sets specified arrays to specified constants in a given flux, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>FLUXNUM</td>
<td>Defines flux regions, for editing data with ADDREG, EQUALREG, MULTIREG and COPYREG, and for inter-region multipliers MULTREGT, MULTREGD and MULTREGH.</td>
</tr>
<tr>
<td>FLUXREG</td>
<td>Select which flux regions are active in a run.</td>
</tr>
<tr>
<td>GRIDFILE</td>
<td>Controls the contents of the Grid Geometry file (for graphical output).</td>
</tr>
<tr>
<td>IMPORT</td>
<td>Imports binary data generated by a grid pre-processor.</td>
</tr>
<tr>
<td>INIT</td>
<td>Produces initial file for GRAF.</td>
</tr>
<tr>
<td>LGRCOPY</td>
<td>Copy global values to local cells before operations on cells within an LGR.</td>
</tr>
<tr>
<td>MAXVALUE</td>
<td>Applies a maximum value to specified arrays within the current box.</td>
</tr>
<tr>
<td>MINNNCT</td>
<td>Sets a cut-off transmissibility below which an NNC is deleted.</td>
</tr>
<tr>
<td>MINVALUE</td>
<td>Applies a minimum value to specified arrays within the current box.</td>
</tr>
<tr>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>MULTIPLY</td>
<td>Multiply specified arrays by specified constants within the current box.</td>
</tr>
<tr>
<td>MULTIREG</td>
<td>Multiply specified arrays by specified constants in a given flux, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>NOGGF</td>
<td>Specifies that no Grid Geometry file is to be produced (for graphical output).</td>
</tr>
<tr>
<td>OPERATE</td>
<td>Performs arithmetic operations on arrays.</td>
</tr>
<tr>
<td>OPERATER</td>
<td>Performs arithmetic operations on arrays for a specific region of the grid.</td>
</tr>
<tr>
<td>OPERNUM</td>
<td>Defines region on which to perform operations using the OPERATER keyword. The data can also be edited using ADDREG, EQUALREG, MULTIREG and COPYREG.</td>
</tr>
<tr>
<td>RPTGRID</td>
<td>Sets report levels for GRID data.</td>
</tr>
<tr>
<td>RPTINIT</td>
<td>Controls output to the Initialization file for GRID and EDIT data.</td>
</tr>
<tr>
<td>SOLVDIRS</td>
<td>Overrides default settings for principal directions in the linear solver.</td>
</tr>
<tr>
<td>SPECGRID</td>
<td>Specification of grid dimensions.</td>
</tr>
<tr>
<td>TOPS</td>
<td>Specifies the depths of the top face of each grid cell.</td>
</tr>
<tr>
<td>USEFLUX</td>
<td>Use a Flux file.</td>
</tr>
</tbody>
</table>

**ECLIPSE 300 only keywords**

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CIRCLE</td>
<td>Requests completion of the circle</td>
</tr>
<tr>
<td>MAXTRANZ</td>
<td>Maximum Z-direction transmissibility.</td>
</tr>
<tr>
<td>MIDS</td>
<td>Specify cell mid-point depths.</td>
</tr>
<tr>
<td>MINDZNET</td>
<td>Minimum net thickness for active cell.</td>
</tr>
<tr>
<td>XSTC</td>
<td>Specifies oil composition at standard conditions for the Flux option.</td>
</tr>
<tr>
<td>YSTC</td>
<td>Specifies gas composition at standard conditions for the Flux option.</td>
</tr>
</tbody>
</table>

**Local grid refinement**

**Common keywords**

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CARFIN</td>
<td>Defines a Cartesian local grid refinement.</td>
</tr>
<tr>
<td>COARSEN</td>
<td>Specifies a box of cells for grid coarsening.</td>
</tr>
<tr>
<td>ENDFIN</td>
<td>Terminates data for local grid refinement.</td>
</tr>
<tr>
<td>HRFIN</td>
<td>Local grid size ratios in the R-direction.</td>
</tr>
<tr>
<td>HXFIN</td>
<td>Local grid size ratios in the X-direction.</td>
</tr>
<tr>
<td>HYFIN</td>
<td>Local grid size ratios in the Y-direction.</td>
</tr>
<tr>
<td>HZFIN</td>
<td>Local grid size ratios in the Z-direction.</td>
</tr>
<tr>
<td>NXFIN</td>
<td>Number of local cells in each global cell of an LGR in the X-direction.</td>
</tr>
<tr>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>---------</td>
<td>-------------</td>
</tr>
<tr>
<td>NYFIN</td>
<td>Number of local cells in each global cell of an LGR in the Y-direction.</td>
</tr>
<tr>
<td>NZFIN</td>
<td>Number of local cells in each global cell of an LGR in the Z-direction.</td>
</tr>
<tr>
<td>RADFIN</td>
<td>Specifies a single-column radial local grid refinement.</td>
</tr>
<tr>
<td>REFINE</td>
<td>Initiates data input for a named local grid.</td>
</tr>
<tr>
<td>RPTGRID</td>
<td>Controls output of GRID section data for local grids.</td>
</tr>
</tbody>
</table>

### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AMALGAM</td>
<td>Allow local grids to amalgamate.</td>
</tr>
<tr>
<td>AUTOCOAR</td>
<td>Specifies a box of cells for coarsening within an auto refinement.</td>
</tr>
<tr>
<td>QMOBIL</td>
<td>Controls the mobile end-point correction in an LGR.</td>
</tr>
<tr>
<td>RADFIN4</td>
<td>Specifies a four-column radial local grid refinement</td>
</tr>
<tr>
<td>RPTGRID</td>
<td>Controls output of GRID section data for local grids.</td>
</tr>
<tr>
<td>SMULTX, SMULTY, SMULTZ</td>
<td>Alternative transmissibility multipliers for auto refinement.</td>
</tr>
<tr>
<td>VEFIN</td>
<td>Controls Vertical Equilibrium in local grids.</td>
</tr>
</tbody>
</table>

### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NOCIRCLE</td>
<td>Turn off completion of circle for local radial refinements.</td>
</tr>
</tbody>
</table>

### Tensor permeability

#### Common keywords

- **MPFANUM** Identifies regions in which to apply the multipoint discretization or nine point scheme.
- **MPFINNC** Sets multi-point transmissibility coefficients.

### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GADJUST</td>
<td>Adjusts diagonal permeability tensor.</td>
</tr>
<tr>
<td>MPFA</td>
<td>Sets options for tensor permeability.</td>
</tr>
<tr>
<td>PERMXX</td>
<td>X-direction tensor permeability coefficient for the current input box.</td>
</tr>
<tr>
<td>PERMYY</td>
<td>Y-direction tensor permeability coefficient for the current input box.</td>
</tr>
<tr>
<td>PERMZZ</td>
<td>Z-direction tensor permeability coefficient for the current input box.</td>
</tr>
<tr>
<td>PERMXY</td>
<td>XY-direction tensor permeability coefficient for the current input box</td>
</tr>
<tr>
<td>PERMYZ</td>
<td>YZ-direction tensor permeability coefficient for the current input box.</td>
</tr>
</tbody>
</table>
**ECLIPSE 300 only keyword**  

<table>
<thead>
<tr>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PERMZX</td>
</tr>
<tr>
<td>TBASIS</td>
</tr>
</tbody>
</table>

**Geomechanics (ECLIPSE 300)**

- **BIOTC** Sets the Biot constant for each grid block.
- **FAULTS** Can define a geomechanics fault where slippage is allowed.
- **GLTHEX** Sets a coefficient of linear thermal expansion for each grid block.
- **POISSONR** Sets the Poisson ratio for each grid block.
- **PROPFLOW** Defines parameters to be used with the proponent flowback feature.
- **ROCKDEN** Sets the rock density for each grid block.
- **YOUNGMOD** Sets the elastic Young’s modulus for each grid block.

**Alphabetical list of GRID section keywords**

**Common keywords**

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACTNUM</td>
<td>Identifies active grid blocks.</td>
</tr>
<tr>
<td>ADD</td>
<td>Adds specified constants to specified arrays in the current box.</td>
</tr>
<tr>
<td>ADDREG</td>
<td>Adds a constant to the specified array in a given flux, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>ADDZCORN</td>
<td>Adds a constant to the cell corner point depths array ZCORN.</td>
</tr>
<tr>
<td>AQUCON</td>
<td>Defines a connection between a numerical aquifer and the reservoir.</td>
</tr>
<tr>
<td>AQUNUM</td>
<td>Defines a numerical aquifer.</td>
</tr>
<tr>
<td>BOUNDARY</td>
<td>Defines the section of the grid for which output maps are required.</td>
</tr>
<tr>
<td>BOX</td>
<td>Redefines the current box.</td>
</tr>
<tr>
<td>CARFIN</td>
<td>Defines a Cartesian local grid refinement.</td>
</tr>
<tr>
<td>COARSENC</td>
<td>Specifies a box of cells for grid coarsening.</td>
</tr>
<tr>
<td>COALNUM</td>
<td>Specifies coal regions for the Coal bed Methane option.</td>
</tr>
<tr>
<td>COORD</td>
<td>Defines the lines that contain all grid block corner points for each (I,J) and for each reservoir in the grid.</td>
</tr>
<tr>
<td>COORDSYS</td>
<td>Specifies completion of the circle in both corner point and block-centered geometries. Allocates grid block layers to reservoirs if there is more than one reservoir in the grid.</td>
</tr>
<tr>
<td>COPY</td>
<td>Copies data in the current box from one specified array to another.</td>
</tr>
<tr>
<td>COPYBOX</td>
<td>Copies data from one box of cells to another box of the same size.</td>
</tr>
<tr>
<td>COPYREG</td>
<td>Copy by FLXNUM region, MULTNUM region or OPERNUM region.</td>
</tr>
<tr>
<td>DIFFMMF</td>
<td>Matrix-fracture diffusivity multipliers for current input box.</td>
</tr>
<tr>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>DIFFMR</td>
<td>R-direction diffusivity multipliers for current input box.</td>
</tr>
<tr>
<td>DIFFMR-</td>
<td>Negative R-direction diffusivity multipliers for current input box.</td>
</tr>
<tr>
<td>DIFFMTH</td>
<td>Theta-direction diffusivity multipliers for current input box.</td>
</tr>
<tr>
<td>DIFFMTH-</td>
<td>Negative Theta-direction diffusivity multipliers for current input box.</td>
</tr>
<tr>
<td>DIFFMX</td>
<td>X-direction diffusivity multipliers for current input box.</td>
</tr>
<tr>
<td>DIFFMX-</td>
<td>Negative X-direction diffusivity multipliers for current input box.</td>
</tr>
<tr>
<td>DIFFMY</td>
<td>Y-direction diffusivity multipliers for current input box.</td>
</tr>
<tr>
<td>DIFFMY-</td>
<td>Negative Y-direction diffusivity multipliers for current input box.</td>
</tr>
<tr>
<td>DIFFMZ</td>
<td>Z-direction diffusivity multipliers for current input box.</td>
</tr>
<tr>
<td>DIFFMZ-</td>
<td>Negative Z-direction diffusivity multipliers for current input box.</td>
</tr>
<tr>
<td>DPGRID</td>
<td>Enables you to enter grid data for matrix cells only.</td>
</tr>
<tr>
<td>DPNUM</td>
<td>Identifies single porosity regions in a dual porosity run.</td>
</tr>
<tr>
<td>DR</td>
<td>R-direction grid block sizes for the current box.</td>
</tr>
<tr>
<td>DRV</td>
<td>Vector of R-direction grid block sizes.</td>
</tr>
<tr>
<td>DTHETA</td>
<td>Theta-direction grid block sizes for the current box.</td>
</tr>
<tr>
<td>DTHETAV</td>
<td>Vector of theta-direction grid block sizes.</td>
</tr>
<tr>
<td>DUMPFLUX</td>
<td>Output a Flux file.</td>
</tr>
<tr>
<td>DX</td>
<td>X-direction grid block sizes for the current box.</td>
</tr>
<tr>
<td>DXV</td>
<td>Vector of X-direction grid block sizes.</td>
</tr>
<tr>
<td>DY</td>
<td>Y-direction grid block sizes for the current box.</td>
</tr>
<tr>
<td>DYV</td>
<td>Vector of Y-direction grid block sizes.</td>
</tr>
<tr>
<td>DZ</td>
<td>Z-direction (vertical) grid block sizes for the current box.</td>
</tr>
<tr>
<td>DZMTRX</td>
<td>The vertical dimension of a typical block of matrix material, for the entire grid.</td>
</tr>
<tr>
<td>DZMTRXV</td>
<td>The vertical dimension of a typical block of matrix material, for the current box.</td>
</tr>
<tr>
<td>DZNET</td>
<td>Grid block net thicknesses for the current box.</td>
</tr>
<tr>
<td>ENDBOX</td>
<td>Redefines the current box to encompass the whole grid.</td>
</tr>
<tr>
<td>ENDFIN</td>
<td>Terminates data for a local grid refinement.</td>
</tr>
<tr>
<td>EQLZCORN</td>
<td>Resets part of the corner point depth array.</td>
</tr>
<tr>
<td>EQUALREG</td>
<td>Sets a specified array to a constant value in a given flux, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>EQUALS</td>
<td>Sets specified arrays to specified constants in the current box.</td>
</tr>
<tr>
<td>EXTFIN</td>
<td>Specifies an unstructured local grid refinement.</td>
</tr>
<tr>
<td>EXTHOST</td>
<td>Specifies the global host cells of an unstructured local grid refinement.</td>
</tr>
<tr>
<td>EXTREPGGL</td>
<td>Specifies the replaced global cells for an unstructured local grid refinement.</td>
</tr>
<tr>
<td>FAULTS</td>
<td>Identifies fault trajectories for subsequent use by MULTFLT keyword.</td>
</tr>
<tr>
<td>FILEUNIT</td>
<td>Confirms the units of the data set.</td>
</tr>
<tr>
<td>FLUXNUM</td>
<td>Defines flux regions, for editing GRID section data.</td>
</tr>
<tr>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>FLUXREG</td>
<td>Select which flux regions are active in a run.</td>
</tr>
<tr>
<td>GDFILE</td>
<td>Inputs corner point positions for each cell from a Grid file.</td>
</tr>
<tr>
<td>GDORIENT</td>
<td>Indicates the orientation and handedness of the grid</td>
</tr>
<tr>
<td>GRIDFILE</td>
<td>Controls contents of grid geometry file (for graphical output).</td>
</tr>
<tr>
<td>GRIDUNIT</td>
<td>Specifies the grid data units that can be output by grid pre-processors or by ECLIPSE simulators writing the grid geometry files</td>
</tr>
<tr>
<td>HMMULTPV</td>
<td>Specifies pore volume multipliers (cumulative).</td>
</tr>
<tr>
<td>HMMULTxx</td>
<td>Specify history matching multipliers on transmissibility and/or permeability.</td>
</tr>
<tr>
<td>HRFIN</td>
<td>Local grid size ratios in the R-direction.</td>
</tr>
<tr>
<td>HXFIN</td>
<td>Local grid size ratios in the X-direction.</td>
</tr>
<tr>
<td>HYFIN</td>
<td>Local grid size ratios in the Y-direction.</td>
</tr>
<tr>
<td>HZFIN</td>
<td>Local grid size ratios in the Z-direction.</td>
</tr>
<tr>
<td>IMPORT</td>
<td>Imports binary data generated by a grid pre-processor.</td>
</tr>
<tr>
<td>INIT</td>
<td>Produces Initial file for GRAF.</td>
</tr>
<tr>
<td>INRAD</td>
<td>Sets inner radius of radial grid.</td>
</tr>
<tr>
<td>JFUNC</td>
<td>Activates Leverett J-Function option for scaling capillary pressure.</td>
</tr>
<tr>
<td>JFUNCR</td>
<td>Activates Leverett J-Function option for scaling capillary pressure on a per saturation table basis.</td>
</tr>
<tr>
<td>LGRCOPY</td>
<td>Copy global values to local cells before operations on cells within an LGR.</td>
</tr>
<tr>
<td>MAPAXES</td>
<td>Inputs map origin from a grid pre-processor, for output to the Grid file.</td>
</tr>
<tr>
<td>MAPUNITS</td>
<td>Specifies the units used for MAPAXES data.</td>
</tr>
<tr>
<td>MAXVALUE</td>
<td>Applies a maximum value to specified arrays within the current box.</td>
</tr>
<tr>
<td>MINPV</td>
<td>Sets a minimum pore volume that each cell must have to be active.</td>
</tr>
<tr>
<td>MINPVV</td>
<td>Sets a vector of minimum pore volumes that the cells in the current box must have to be active.</td>
</tr>
<tr>
<td>MINVALUE</td>
<td>Applies a minimum value to specified arrays within the current box.</td>
</tr>
<tr>
<td>MPFANUM</td>
<td>Identifies regions in which to apply the multipoint discretization or nine point scheme.</td>
</tr>
<tr>
<td>MPPNCC</td>
<td>Sets multi-point transmissibility coefficients.</td>
</tr>
<tr>
<td>MULTFLT</td>
<td>Multipliers for the transmissibility and diffusivity across faults defined by the FAULTS keyword.</td>
</tr>
<tr>
<td>MULTIPLY</td>
<td>Multiplies specified arrays by constant values in the current box.</td>
</tr>
<tr>
<td>MULTIREG</td>
<td>Multiplies specified arrays by constant values in a flux, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>MULTNUM</td>
<td>Identifies multiplier regions for use with MULTREGT, MULTREGD and MULTREGH. Also defines regions, for editing data with ADDREG, EQUALREG, MULTIREG and COPYREG.</td>
</tr>
<tr>
<td>MULTPV</td>
<td>Pore volume multipliers for the current box</td>
</tr>
<tr>
<td>MULTR</td>
<td>R-direction transmissibility multipliers for the current box.</td>
</tr>
<tr>
<td>MULTR−</td>
<td>Negative R-direction transmissibility multipliers for the current box.</td>
</tr>
<tr>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>MULTREGP</td>
<td>Multiplies pore volumes between flux or multiplier regions.</td>
</tr>
<tr>
<td>MULTREGT</td>
<td>Multiplies transmissibilities between flux or multiplier regions.</td>
</tr>
<tr>
<td>MULTTHT</td>
<td>Theta-direction transmissibility multipliers for the current box.</td>
</tr>
<tr>
<td>MULTTHT-</td>
<td>Negative R-direction transmissibility multipliers for the current box.</td>
</tr>
<tr>
<td>MULTX</td>
<td>X-direction transmissibility multipliers for the current box.</td>
</tr>
<tr>
<td>MULTX-</td>
<td>Negative X-direction transmissibility multipliers for the current box.</td>
</tr>
<tr>
<td>MULTY</td>
<td>Y-direction transmissibility multipliers for the current box.</td>
</tr>
<tr>
<td>MULTY-</td>
<td>Negative Y-direction transmissibility multipliers for the current box.</td>
</tr>
<tr>
<td>MULTZ</td>
<td>Z-direction transmissibility multipliers for the current box.</td>
</tr>
<tr>
<td>MULTZ-</td>
<td>Negative Z-direction transmissibility multipliers for the current box.</td>
</tr>
<tr>
<td>NNC</td>
<td>Explicit entry of non-neighbor connections.</td>
</tr>
<tr>
<td>NOGGF</td>
<td>Stops production of grid geometry file (for graphical output).</td>
</tr>
<tr>
<td>NODPPM</td>
<td>Specifies that permeabilities in the fracture cells are not to be multiplied by the fracture porosity.</td>
</tr>
<tr>
<td>NTG</td>
<td>Grid block net-to-gross ratios for the current box.</td>
</tr>
<tr>
<td>NXFIN</td>
<td>Number of local cells in each global cell of an LGR in the X-direction.</td>
</tr>
<tr>
<td>NYFIN</td>
<td>Number of local cells in each global cell of an LGR in the Y-direction.</td>
</tr>
<tr>
<td>NZFIN</td>
<td>Number of local cells in each global cell of an LGR in the Z-direction.</td>
</tr>
<tr>
<td>OPERATE</td>
<td>Performs arithmetic operations on arrays.</td>
</tr>
<tr>
<td>OPERATER</td>
<td>Performs arithmetic operations on arrays for a specific region of the grid.</td>
</tr>
<tr>
<td>OPERNUM</td>
<td>Defines region on which to perform operations using the OPERATER keyword. The data can also be edited using ADDREG, EQUALREG, MULTIREG and COPYREG.</td>
</tr>
<tr>
<td>PERMR</td>
<td>R-direction permeabilities for the current box.</td>
</tr>
<tr>
<td>PERMTHT</td>
<td>Theta-direction permeabilities for the current box.</td>
</tr>
<tr>
<td>PERMX</td>
<td>X-direction permeabilities for the current box.</td>
</tr>
<tr>
<td>PERMY</td>
<td>Y-direction permeabilities for the current box.</td>
</tr>
<tr>
<td>PERMZ</td>
<td>Z-direction permeabilities for the current box.</td>
</tr>
<tr>
<td>PINCH</td>
<td>Generates connections across pinched-out layers.</td>
</tr>
<tr>
<td>PINCHOUT</td>
<td>Same as keyword PINCH, but threshold thickness cannot be adjusted.</td>
</tr>
<tr>
<td>PINCHXY</td>
<td>Generates horizontal (I, J) pinchout connections.</td>
</tr>
<tr>
<td>PORO</td>
<td>Grid block porosities for the current box.</td>
</tr>
<tr>
<td>RADFIN</td>
<td>Specifies a single-column radial local grid refinement.</td>
</tr>
<tr>
<td>REFINE</td>
<td>Initiates data input for a named local grid.</td>
</tr>
<tr>
<td>RESVNUM</td>
<td>Initiates COORD data input for a given reservoir in the grid.</td>
</tr>
<tr>
<td>ROCKFRAC</td>
<td>Sets the rock volume fraction for each grid block.</td>
</tr>
<tr>
<td>RPTGRID</td>
<td>Sets report levels for GRID data.</td>
</tr>
<tr>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>RPTINIT</td>
<td>Controls output to the Initialization file for GRID and EDIT data.</td>
</tr>
<tr>
<td>RPTGRIDL</td>
<td>Controls output of GRID section data for local grids.</td>
</tr>
<tr>
<td>SIGMA</td>
<td>Matrix-to-fracture coupling factor for the entire grid.</td>
</tr>
<tr>
<td>SIGMAGD</td>
<td>Matrix-fracture coupling for oil-gas gravity drainage (whole grid).</td>
</tr>
<tr>
<td>SIGMAGDV</td>
<td>Matrix-fracture coupling for oil-gas gravity drainage (current box).</td>
</tr>
<tr>
<td>SIGMAV</td>
<td>Matrix-to-fracture coupling factor for the current box.</td>
</tr>
<tr>
<td>SOLVDIMS</td>
<td>Specifies the solver dimensions for an unstructured local grid refinement.</td>
</tr>
<tr>
<td>SOLVDIRS</td>
<td>Overrides default settings for principal directions in the linear solver.</td>
</tr>
<tr>
<td>SOLVNUM</td>
<td>Specifies the map from user numbers to solver numbers for an unstructured grid.</td>
</tr>
<tr>
<td>SPECGRID</td>
<td>Specification of grid dimensions and characteristics.</td>
</tr>
<tr>
<td>THCONR</td>
<td>Rock and fluid thermal conductivities for the current box, for the Temperature and Thermal options.</td>
</tr>
<tr>
<td>THPRESFT</td>
<td>Specifies a threshold pressure associated with a named fault.</td>
</tr>
<tr>
<td>TOPS</td>
<td>Depths of top faces of grid blocks for the current box.</td>
</tr>
<tr>
<td>TRANGL</td>
<td>Specifies the global-local connections and associated transmissibilities for an unstructured local grid refinement.</td>
</tr>
<tr>
<td>USEFLUX</td>
<td>Use a Flux file.</td>
</tr>
<tr>
<td>ZCORN</td>
<td>Depths of grid block corners.</td>
</tr>
</tbody>
</table>

### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>AMALGAM</td>
<td>Allow local grids to amalgamate.</td>
</tr>
<tr>
<td></td>
<td>AQUUNNC</td>
<td>Specifies numerical aquifer non-neighbor connection properties explicitly.</td>
</tr>
<tr>
<td></td>
<td>AUTOCOAR</td>
<td>Specifies a box of cells for coarsening within an auto refinement</td>
</tr>
<tr>
<td></td>
<td>COLLAPSE</td>
<td>Specifies which cells should be collapsed in Compressed VE option.</td>
</tr>
<tr>
<td></td>
<td>HALFTRAN</td>
<td>Activates the half-block transmissibility upscaling option</td>
</tr>
<tr>
<td></td>
<td>HMAQUNUM</td>
<td>Activates the calculation of gradients for numerical aquifer parameters.</td>
</tr>
<tr>
<td></td>
<td>HMMLAQUN</td>
<td>Specifies cumulative multipliers on numerical aquifer parameters.</td>
</tr>
<tr>
<td></td>
<td>HMMMREGT</td>
<td>Specifies cumulative inter-region transmissibility multipliers</td>
</tr>
<tr>
<td></td>
<td>HMMULRGT</td>
<td>Activates the calculation of gradients with respect to inter-region transmissibility multipliers.</td>
</tr>
<tr>
<td></td>
<td>IONROCK</td>
<td>Rock ion exchange capacity</td>
</tr>
<tr>
<td></td>
<td>ISOLNUM</td>
<td>Defines the extent of each independent reservoir region.</td>
</tr>
<tr>
<td></td>
<td>LINKPERM</td>
<td>Treats grid block permeabilities as applying to cell faces.</td>
</tr>
<tr>
<td></td>
<td>LTOSIGMA</td>
<td>Provides an easy way to calculate SIGMA from LX, LY, LZ.</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 100 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-------------------</td>
<td>--------------------------</td>
<td>----------------------------------------------------------------------------</td>
</tr>
<tr>
<td></td>
<td>LX, LY, LZ</td>
<td>Inputs the representative matrix block sizes for the dual porosity viscous displacement option.</td>
</tr>
<tr>
<td></td>
<td>MULTREGD</td>
<td>Multiplies diffusivities between flux or multiplier regions.</td>
</tr>
<tr>
<td></td>
<td>MULTREGH</td>
<td>Multiplies thermal conductivity between flux or multiplier regions.</td>
</tr>
<tr>
<td>Do not use with block centered geometry.</td>
<td>NEWTRAN</td>
<td>Specifies that transmissibilities are to be computed using area of contact between blocks. Default for corner point geometry.</td>
</tr>
<tr>
<td></td>
<td>NINENUM</td>
<td>Specifies the extent of the nine-point discretization scheme.</td>
</tr>
<tr>
<td></td>
<td>NMATOPTS</td>
<td>Controls the discretized matrix Dual Porosity option.</td>
</tr>
<tr>
<td></td>
<td>OLDTRAN</td>
<td>Specifies that block center data should be used to compute transmissibilities.</td>
</tr>
<tr>
<td></td>
<td>OLDTRANR</td>
<td>Alternative calculation of transmissibilities using block center data.</td>
</tr>
<tr>
<td></td>
<td>OUTRAD</td>
<td>Sets outer radius of radial grid.</td>
</tr>
<tr>
<td></td>
<td>PEBI</td>
<td>Specifies that an unstructured grid is being used.</td>
</tr>
<tr>
<td></td>
<td>PERMAVE</td>
<td>Sets options for permeability averaging in transmissibilities.</td>
</tr>
<tr>
<td></td>
<td>PINCHNUM</td>
<td>Identifies pinchout regions for use by PINCHREG</td>
</tr>
<tr>
<td></td>
<td>PINCHREG</td>
<td>Specifies pinchout data within PINCHNUM regions.</td>
</tr>
<tr>
<td></td>
<td>QMOBIL</td>
<td>Controls the mobile end-point correction in an LGR.</td>
</tr>
<tr>
<td></td>
<td>RADFIN4</td>
<td>Specifies a four-column radial local grid refinement.</td>
</tr>
<tr>
<td></td>
<td>RPTISOL</td>
<td>Generates data for the ISOLNUM keyword, output to the Debug file.</td>
</tr>
<tr>
<td></td>
<td>SMULTX, SMULTY, SMULTZ</td>
<td>Alternative transmissibility multipliers for auto refinement.</td>
</tr>
<tr>
<td></td>
<td>VEDDEBUG</td>
<td>Controls debug output from the Vertical Equilibrium options.</td>
</tr>
<tr>
<td></td>
<td>VEFIN</td>
<td>Controls Vertical Equilibrium in local grids.</td>
</tr>
</tbody>
</table>

**ECLIPSE 300 only keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>AQUANCON</td>
<td>Defines connection data for analytic aquifer</td>
</tr>
<tr>
<td></td>
<td>AQUCT</td>
<td>Specifies property data for Cater-Tracy aquifer</td>
</tr>
<tr>
<td></td>
<td>BIOTC</td>
<td>Sets the Biot constant for each grid block</td>
</tr>
<tr>
<td>Radial geometry only</td>
<td>CIRCLE</td>
<td>Requests completion of circle</td>
</tr>
<tr>
<td></td>
<td>COALNUMR</td>
<td>Redefine coal region numbers in connection with the multi-porosity option.</td>
</tr>
<tr>
<td></td>
<td>CONDFLTS</td>
<td>Specifies geometry of conductive faults.</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 300 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-------------</td>
<td>--------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td>CONDFRAC</td>
<td></td>
<td>Specifies geometry of a conductive fracture</td>
</tr>
<tr>
<td>CRNDENS</td>
<td></td>
<td>Specifies rock density for each coal region number</td>
</tr>
<tr>
<td>CRITPERM</td>
<td></td>
<td>Sets a critical permeability for cell collapse in Compressed VE</td>
</tr>
<tr>
<td>DZV</td>
<td></td>
<td>Vector of Z-direction grid block sizes</td>
</tr>
<tr>
<td>GADJUST</td>
<td></td>
<td>Adjusts diagonal permeability tensor</td>
</tr>
<tr>
<td>GLTHEX</td>
<td></td>
<td>Sets a coefficient of linear thermal expansion for each grid block.</td>
</tr>
<tr>
<td>Thermal</td>
<td>HEATCR</td>
<td>Rock heat capacity</td>
</tr>
<tr>
<td>Thermal</td>
<td>HEATCRT</td>
<td>Temperature dependence of rock heat capacity</td>
</tr>
<tr>
<td>MAXTRANZ</td>
<td></td>
<td>Maximum Z-direction transmissibility.</td>
</tr>
<tr>
<td>MIDS</td>
<td></td>
<td>Specify cell mid-point depths.</td>
</tr>
<tr>
<td>MIND2NET</td>
<td></td>
<td>Minimum net thickness for active cell.</td>
</tr>
<tr>
<td>MINPORV9</td>
<td></td>
<td>Set a minimum pore volume for calculation of nine-point transmissibilities.</td>
</tr>
<tr>
<td>Thermal</td>
<td>MINROCKV</td>
<td>Sets a minimum rock volume that each cell must have to be active</td>
</tr>
<tr>
<td>MPFA</td>
<td></td>
<td>Sets options for tensor permeability.</td>
</tr>
<tr>
<td>MULTMF</td>
<td></td>
<td>Transmissibility multipliers for matrix-fracture coupling.</td>
</tr>
<tr>
<td>NNGEN</td>
<td></td>
<td>Allows NNCs to be generated between any two grid cells.</td>
</tr>
<tr>
<td>NOCIRCLE</td>
<td></td>
<td>Turn off completion of circle for local grid refinements.</td>
</tr>
<tr>
<td>ORTHERO</td>
<td></td>
<td>Set a minimum orthogonality error for calculation of nine-point transmissibilities.</td>
</tr>
<tr>
<td>ORTHERX</td>
<td></td>
<td>Set a minimum orthogonality error for calculation of nine-point transmissibilities for interfaces in the X-direction.</td>
</tr>
<tr>
<td>ORTHERRY</td>
<td></td>
<td>Set a minimum orthogonality error for calculation of nine-point transmissibilities for interfaces in the Y-direction.</td>
</tr>
<tr>
<td>ORTHERRZ</td>
<td></td>
<td>Set a minimum orthogonality error for calculation of nine-point transmissibilities for interfaces in the Z-direction.</td>
</tr>
<tr>
<td>PERMMF</td>
<td></td>
<td>Permeability for matrix-fracture coupling.</td>
</tr>
<tr>
<td>PERMX</td>
<td></td>
<td>X-direction tensor permeability coefficient for the current input box.</td>
</tr>
<tr>
<td>PERMYY</td>
<td></td>
<td>Y-direction tensor permeability coefficient for the current input box.</td>
</tr>
<tr>
<td>PERMZ</td>
<td></td>
<td>Z-direction tensor permeability coefficient for the current input box.</td>
</tr>
<tr>
<td>PERMYX</td>
<td></td>
<td>XY-direction tensor permeability coefficient for the current input box.</td>
</tr>
<tr>
<td>PERMYZ</td>
<td></td>
<td>YZ-direction tensor permeability coefficient for the current input box.</td>
</tr>
<tr>
<td>PERMZX</td>
<td></td>
<td>ZX-direction tensor permeability coefficient for the current input box.</td>
</tr>
<tr>
<td>POISSONR</td>
<td></td>
<td>Sets the Poisson ratio for each grid block.</td>
</tr>
<tr>
<td>PROPFLOW</td>
<td></td>
<td>Defines parameters to be used with the proponent flowback feature</td>
</tr>
</tbody>
</table>
### Example data set

```
RUNSPEC
TITLE : Example of simple 5x3x2 block centered Cartesian model
DIMENS
-- NX NY NZ
   5 3 2 / -- remainder of RUNSPEC should follow
GRID
   DVX 1500 3*1000 2000 / X dimensions of blocks only varies in X direction
   DVY 3*2000 / -- Set constant values
EQUALS
   DZ 50 1 5 1 3 1 1 / layer 1 DZ
   PERMX 100 / layer 1 PERMX box unchanged
   DZ 75 1 5 1 3 2 2 / layer 2 DZ box now I=1-5, J=1-3, K=2
   PERMX 80 / -- Set individual values for Porosity and Tops for layer 1
          PORO 0.3 / -- End Equals keyword
       BOX 1 5 1 3 1 1 / defines box I=1-5, J=1-3, K=1
          PORO individual values for 15 blocks in the box
             0.2 0.21 0.22 0.23 0.24
             0.21 0.22 0.23 0.26 0.28
             0.19 0.27 0.28 0.25 0.24 /
             TOPS 5*7000 5*7100 5*7200 / ENDBOX
-- Box no longer defined; defaults to full grid I=1-5, J=1-3, K=1-2
COPY
```
EDIT section

The EDIT section contains instructions for modifying the pore volumes, block center depths, transmissibilities, diffusivities (for the Molecular Diffusion option), and non-neighbor connections (NNCs) computed by the program from the data entered in the GRID section.

It is entirely optional.

The GRID output array keywords are:

<table>
<thead>
<tr>
<th>DEPTH</th>
<th>PORV</th>
<th>TRANX</th>
<th>TRANR</th>
<th>DIFFX</th>
<th>DIFFR</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRANY</td>
<td>TRANHT</td>
<td>DIFFY</td>
<td>DIFFHT</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TRANZ</td>
<td>DIFFZ</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

These keywords may be used in the EDIT section to overwrite data either for the entire reservoir, or for a set of the grid blocks defined using the BOX keyword.

Alternatively, the multiplier keywords:

MULTX, MULTR
MULTY, MULTTHT
MULTZ, MULTPV

_ECLIPSE 100_ MULTX-, MULTR-
_ECLIPSE 100_ MULTY-, MULTTHT-
_ECLIPSE 100_ MULTZ-

and the operational keywords:

<table>
<thead>
<tr>
<th>BOX</th>
<th>EQUALS</th>
<th>ADD</th>
</tr>
</thead>
<tbody>
<tr>
<td>ENDBOX</td>
<td>COPY</td>
<td>MULTIPLY</td>
</tr>
<tr>
<td>MINVALUE</td>
<td>MAXVALUE</td>
<td></td>
</tr>
</tbody>
</table>

may be used to modify the arrays exactly as in the GRID section. In addition, the MULTFLT keyword may be used to modify the transmissibility across faults entered using the keyword FAULTS in the GRID section.
Changing cell depths

The cell center depths obtained in the GRID section calculations are used to obtain hydrostatic pressure differences between cells during the simulation. These may be changed by using the DEPTH keyword to set depths for all the cells, or ADD, COPY, EQUALS and MULTIPLY to alter boxes of values. Note that changing the depths does not alter the plotted grid.

Changing pore volumes

As in the case of DEPTH, the calculated pore volumes may be changed. The PORV keyword may be used to set new values for all the cells, or EQUALS, ADD and MULTIPLY to alter boxes of values.

*Note:* If the pore volume value is added to a cell that is inactive, it remains inactive unless the ACTNUM array is also modified.

*Note:* If a cell is defined with zero pore volume in the GRID section, then making it active by changing the pore volume in the EDIT section using PORV, is not allowed with the polymer, surfactant, foam, temperature or environmental tracer options.

Changing transmissibilities

The keywords to overwrite transmissibility values depend upon the use of the radial or Cartesian geometry options. The program accepts the radial form in a Cartesian run and vice versa, but issues a warning.

<table>
<thead>
<tr>
<th>Cartesian Keyword</th>
<th>Radial Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRANX</td>
<td>TRANR</td>
<td>X or R direction transmissibilities</td>
</tr>
<tr>
<td>TRANY</td>
<td>TRANTHT</td>
<td>Y or θ transmissibilities</td>
</tr>
<tr>
<td>TRANZ</td>
<td>TRANZ</td>
<td>Z transmissibilities</td>
</tr>
</tbody>
</table>

*Table 2.6: Keywords used to change transmissibility values*

As in the case of PORV and DEPTH, these keywords may be specified with values for the whole field, or used through ADD, COPY, EQUALS and MULTIPLY keywords.

Changing diffusivities

If diffusion is being modeled, the DIFFUSE keyword having been specified in the RUNSPEC section, the DIFFX, DIFFY and DIFFZ keywords may be used in the same way as the TRANX, TRANY and TRANZ keywords to overwrite the calculated diffusivities (or DIFFR, DIFFHT and DIFFZ in the radial case).

Diffusivities reflect the effect of cell properties upon diffusive flow, and their evaluation is detailed in “Diffusion” in the ECLIPSE Technical Description.

Modifying non-neighbor connections

The keyword EDITNNC may be used to modify any non-neighbor connections input directly or generated automatically in the GRID section. In other words, this keyword enables the non-neighbor connection transmissibilities (and diffusivities) to be edited directly.
Alternatively, you may wish to use keyword `EDITNNCR` to replace the non-neighbor connection values completely, rather than multiplying the existing ones.

**Note:** `TRANX` and related keywords do not change the non-neighbor connection transmissibilities. However, using `MULTX` and related keywords in the `GRID` section do change the transmissibilities, both between neighboring blocks and along automatically-generated non-neighbor connections, from the grid block in the positive direction.

**Note:** Similarly, when using the molecular diffusion option, `DIFFX` and similar keywords do not change non-neighbor connection diffusivities, but `DIFFMX` and similar keywords used in the `GRID` section do change the diffusivities in the automatically-generated non-neighbor connections in the positive direction.

If `MULTX` and similar keywords are specified in the `EDIT` section, then they act as a multipliers on the edited data. The transmissibility of any NNCs are adjusted just as if the additional multiplier were specified in the `GRID` section.

The keyword `MULTFLT` provides a way of adjusting the transmissibility multipliers in the `EDIT` section, and the NNC transmissibilities across the fault are edited automatically.

**Note:** Use of multiplier keywords in the `EDIT` section is not recommended in general, except as modifiers to transmissibilities entered explicitly using `TRANX` and related keywords in the `EDIT` section. Transmissibility multipliers should normally be specified only in the `GRID` section. The multiplier keywords should not be used in the `EDIT` section when the Local Grid Refinement option is employed.

**Output control**

Output of pore volumes, block center depths, transmissibilities, diffusivities and NNCs is controlled by switches or mnemonics following the `RPTGRID` keyword in the `GRID` section.

However, the values that are printed include the effect of any changes made in the `EDIT` section.

**Alphabetical list of EDIT section keywords**

**Common keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADD</td>
<td>Adds specified constants to specified arrays in the current input box.</td>
<td></td>
</tr>
<tr>
<td>ADDREG</td>
<td>Adds a constant to the specified array in a flux, <code>MULTNUM</code> or <code>OPERNUM</code> region.</td>
<td></td>
</tr>
<tr>
<td>BOUNDARY</td>
<td>Defines the section of the grid for which output maps are required.</td>
<td></td>
</tr>
<tr>
<td>BOX</td>
<td>Re-defines the current input box. <code>COPY</code> Copies data in the current box from one specified array to another.</td>
<td></td>
</tr>
<tr>
<td>COPYREG</td>
<td>Copy by <code>FLUXNUM</code> region <code>MULTNUM</code> region or <code>OPERNUM</code> region.</td>
<td></td>
</tr>
<tr>
<td>DEPTH</td>
<td>Grid block center depths for the current input box.</td>
<td></td>
</tr>
<tr>
<td>Restriction</td>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>----------------------------------------------------------------------------</td>
<td>---------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Molecular Diffusion option in radial geometry only</td>
<td>DIFFR</td>
<td>R-direction diffusivities for the current input box</td>
</tr>
<tr>
<td>Molecular Diffusion option in Cartesian geometry only</td>
<td>DIFFHT</td>
<td>Theta-direction diffusivities for the current input box</td>
</tr>
<tr>
<td>Molecular Diffusion option in Cartesian geometry only</td>
<td>DIFFX</td>
<td>X-direction diffusivities for the current input box</td>
</tr>
<tr>
<td>Molecular Diffusion option in Cartesian geometry only</td>
<td>DIFFY</td>
<td>Y-direction diffusivities for the current input box</td>
</tr>
<tr>
<td>Molecular Diffusion option in radial or Cartesian geometry</td>
<td>DIFFZ</td>
<td>Z-direction diffusivities for the current input box</td>
</tr>
<tr>
<td></td>
<td>EDITNCC</td>
<td>Modifies non-neighbor connections. <strong>EDITNNCR</strong> Replaces non-neighbor connections.</td>
</tr>
<tr>
<td></td>
<td>ENDBOX</td>
<td>Re-defines the current input box to encompass the whole grid.</td>
</tr>
<tr>
<td></td>
<td>ENDFIN</td>
<td>Terminates data for a local grid refinement.</td>
</tr>
<tr>
<td></td>
<td>EQUALREG</td>
<td>Sets an array to a constant value in a flux, MULTNUM, or OPERNUM region.</td>
</tr>
<tr>
<td></td>
<td>EQUALS</td>
<td>Sets specified arrays to specified constants within the current box.</td>
</tr>
<tr>
<td></td>
<td>FILEUNIT</td>
<td>Confirms the units of the data set.</td>
</tr>
<tr>
<td></td>
<td>HMMULTPV</td>
<td>History matching pore volume multipliers.</td>
</tr>
<tr>
<td></td>
<td>HMMULTxx</td>
<td>History matching transmissibility multipliers.</td>
</tr>
<tr>
<td></td>
<td>IMPORT</td>
<td>Imports binary data generated by a grid pre-processor.</td>
</tr>
<tr>
<td></td>
<td>LGRCOPY</td>
<td>Copy global values to local cells before operations on cells within an LGR.</td>
</tr>
<tr>
<td></td>
<td>MAXVALUE</td>
<td>Applies a maximum value to specified arrays within the current box.</td>
</tr>
<tr>
<td></td>
<td>MINVALUE</td>
<td>Applies a minimum value to specified arrays within the current box.</td>
</tr>
<tr>
<td></td>
<td>MULTFLT</td>
<td>Applies a transmissibility multiplier to a named fault defined by the FAULTS keyword in the GRID section.</td>
</tr>
<tr>
<td></td>
<td>MULTIPLY</td>
<td>Multiplies specified arrays by specified constants within the current box.</td>
</tr>
<tr>
<td></td>
<td>MULTIREG</td>
<td>Multiplies an array by a constant in a given flux, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td></td>
<td>MULTPV</td>
<td>Multiplies the calculated pore volume within the current box.</td>
</tr>
<tr>
<td>Restriction</td>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-------------</td>
<td>-----------</td>
<td>------------------------------------------------------------------------------</td>
</tr>
<tr>
<td></td>
<td>MULTREGT</td>
<td>Multiplies transmissibilities between flux or multiplier regions.</td>
</tr>
<tr>
<td></td>
<td>MULTREGP</td>
<td>Multiplies pore volumes between flux or multiplier regions.</td>
</tr>
<tr>
<td></td>
<td>OPERATE</td>
<td>Performs arithmetic operations on arrays.</td>
</tr>
<tr>
<td></td>
<td>OPERATER</td>
<td>Performs arithmetic operations on arrays for a specific region of the grid.</td>
</tr>
<tr>
<td></td>
<td>PORV</td>
<td>Grid block pore volumes for the current input box.</td>
</tr>
<tr>
<td></td>
<td>REFINE</td>
<td>Select local grid refinement for input data.</td>
</tr>
<tr>
<td>Thermal</td>
<td>ROCKV</td>
<td>Rock volume</td>
</tr>
<tr>
<td>Radial geometry only</td>
<td>TRANR</td>
<td>R direction transmissibilities for the current input box.</td>
</tr>
<tr>
<td>Radial geometry only</td>
<td>TRANTHT</td>
<td>Theta direction transmissibilities for the current input box.</td>
</tr>
<tr>
<td>Cartesian geometry only</td>
<td>TRANX</td>
<td>X direction transmissibilities for the current input box.</td>
</tr>
<tr>
<td>Cartesian geometry only</td>
<td>TRANY</td>
<td>Y direction transmissibilities for the current input box.</td>
</tr>
<tr>
<td>Radial or Cartesian geometry</td>
<td>TRANZ</td>
<td>Z direction transmissibilities for the current input box.</td>
</tr>
</tbody>
</table>

**ECLIPSE 100 only keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radial geometry only</td>
<td>MULTR</td>
<td>Sets an additional R-direction transmissibility multiplier within the current box.</td>
</tr>
<tr>
<td>Radial geometry only</td>
<td>MULTR-</td>
<td>Sets an additional negative R-direction transmissibility multiplier within the current box.</td>
</tr>
<tr>
<td></td>
<td>MULTREGD</td>
<td>Multiplies diffusivities between flux or multiplier regions.</td>
</tr>
<tr>
<td></td>
<td>MULTREGH</td>
<td>Multiplies thermal conductivity between flux or multiplier regions.</td>
</tr>
<tr>
<td>Radial geometry only</td>
<td>MULTTHT</td>
<td>Sets an additional Theta-direction transmissibility multiplier within the current box.</td>
</tr>
<tr>
<td>Radial geometry only</td>
<td>MULTTHT-</td>
<td>Sets an additional negative Theta-direction transmissibility multiplier within the current box.</td>
</tr>
<tr>
<td></td>
<td>MULTX</td>
<td>Sets an additional X-direction transmissibility multiplier within the current box.</td>
</tr>
<tr>
<td></td>
<td>MULTX-</td>
<td>Sets an additional negative X-direction transmissibility multiplier within the current box.</td>
</tr>
<tr>
<td></td>
<td>MULTY</td>
<td>Sets an additional Y-direction transmissibility multiplier within the current box.</td>
</tr>
<tr>
<td></td>
<td>MULTY-</td>
<td>Sets an additional negative Y-direction transmissibility multiplier within the current box.</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 100 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-------------</td>
<td>--------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td></td>
<td>MULTZ</td>
<td>Sets an additional Z-direction transmissibility multiplier within the current box.</td>
</tr>
<tr>
<td></td>
<td>MULTZ−</td>
<td>Sets an additional negative Z-direction transmissibility multiplier within the current box.</td>
</tr>
</tbody>
</table>

### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>HEATTR</td>
<td>Heat transmissibility values in the R-direction.</td>
</tr>
<tr>
<td></td>
<td>HEATTHT</td>
<td>Heat transmissibility values in the Theta-direction.</td>
</tr>
<tr>
<td></td>
<td>HEATTX</td>
<td>Heat transmissibility values in the X-direction.</td>
</tr>
<tr>
<td></td>
<td>HEATTY</td>
<td>Heat transmissibility values in the Y-direction.</td>
</tr>
<tr>
<td></td>
<td>HEATTZ</td>
<td>Heat transmissibility values in the Z-direction.</td>
</tr>
<tr>
<td>Thermal</td>
<td>ROCKV</td>
<td>Rock volume</td>
</tr>
</tbody>
</table>

### PROPS section

The PROPS section of the input data contains pressure and saturation dependent properties of the reservoir fluids and rocks.

### Data input

The data is input in multi-tabular keywords, with only one entry of any keyword being accepted. The number of tables of each type is specified in the RUNSPEC section of data. The correct number of tables must be supplied. The RUNSPEC section also specifies the maximum size of each table. When multiple tables are entered after a keyword, each table is terminated by a slash (/).

The keywords in the PROPS section may be specified in any order. All keywords must start in column 1. All characters up to column 8 are significant.

### Example

```plaintext
SWFN
  data for table 1 /
  data for table 2 /
  data for table 3 /
```

### Properties required

The keywords required are determined by whether an ECLIPSE 100 or ECLIPSE 300 Black Oil or an ECLIPSE 300 Compositional Model is used.
The data must always contain the rock compressibility (not strictly required by ECLIPSE 300), and relative permeabilities and capillary pressures as a function of saturation for the phases present.

In a black oil run (ECLIPSE 100 or ECLIPSE 300), the PVT keywords required are determined by the phases selected in the RUNSPEC section (keywords OIL, WATER, GAS, DISGAS, VAPOIL) and any special options selected (keywords APIBRINE, COAL, DIFFUSE, ENDScale, FOAM, MIScIBLE, POLYMER, ROCKCOMP, SATOPTS, TEMP, TRACERS, VE, SURFACT, SOLVENT) in the RUNSPEC section.

In an ECLIPSE 300 compositional run, the PVT keywords required are determined by the EoS (keyword EOS) and any special options (GASWAT, CO2SOL, GASSOL, TRCOEF, MIScIBLE). Specify a black oil run in ECLIPSE 300 by including the BLACKOIL keyword instead of EOS.

The keyword TABDIMS in RUNSPEC may be used to set non-default table sizes (or AQUDIMS, MIScIBLE, ROCKCOMP, TRACERS for the special options).

Saturation functions

Saturation functions are used to describe properties that depend on the phase saturations, namely the relative permeabilities and capillary pressures.

In two-phase systems there is only one independent saturation and the properties are one dimensional functions.

In three-phase systems there are two independent saturations and properties can be either one or two dimensional functions. A number of ways of modeling three phase properties are available:

- Capillary pressures and the water and gas relative permeabilities are treated as one dimensional functions, but the oil relative permeability is treated as a two dimensional function by interpolating the oil-gas and oil-water relative permeabilities. Interpolation of the oil relative permeability is performed using either the default method or one of Stone’s methods.

- Two dimensional tables can be entered for any (or all) of the phase relative permeabilities.

- The Baker methods can be used to describe any (or all) of the phase relative permeabilities as two dimensional functions.

The methods described above can be applied to individual phases, and different methods can be used for each phase. Two further methods are available that apply to all phases:

- The IKU three phase method.

- The ODD3P three phase method.

See "Saturation Functions" in the ECLIPSE Technical Description for further details.

One dimensional saturation functions

There is a choice of two families of keywords for entering relative permeabilities and capillary pressures as functions of a single saturation. The first family: SWOF and (SGOF or SLGOF), allows you to enter the oil relative permeabilities in the same tables as the water and gas relative permeabilities. The second family: SWFN, SGFN and (SOF3 or SOF2), requires the oil relative permeabilities to be entered in a separate table versus oil saturation.

You can select whichever family is best suited to the form of the original data, except that the second family must be used in gas-water runs and runs employing the ECLIPSE 100 Miscible Flood or Solvent options. Keywords from the two families cannot be mixed.
A single saturation function table might be of the form:

```
SWFN
  --Sw    Krw    Pcow
0.22    .0    7.0
0.3     .0    4.0
0.5     .24   2.5
0.8     .65   1.0
0.9     .83    .5
1.0    1.00    .0  /
```

As with all keywords, the data may be spaced or split over lines at will, but it is convenient here to place it into columns. In all but the first column, default values may be entered, and these values are filled in by linear interpolation.

The slash character (/) terminates the table above. If a run has more than one saturation table region, these must be entered in sequence, each terminated by a slash character (/). The tables are assigned to blocks later, in the `REGIONS` section.

Some saturation values in the left-most column are significant:

- $S_w^{co}$: The first value in the table is the connate water, 0.22 in this case, the minimum water saturation in a block in the initial state.
- $S_w^{cr}$: The saturation value at which water becomes mobile is the critical water saturation, 0.3 in this case.

The second column contains water relative permeabilities, and the third the water-oil capillary pressures, defined as $P_{cow} = P_o - P_w$.

The gas saturation functions can be input in a similar way using the `SGFN` keyword. The first value should be zero, as ‘connate’ gas does not exist, and the critical gas is defined in the same manner as the critical water. Gas-oil capillary pressures are defined as $P_{cor} = P_g - P_o$.

In gas-water runs the relative permeability and capillary pressure data can be entered with one keyword, `SGWFN`, `SGFN` and `SWFN`.

### Two dimensional models for saturation functions (for three phase simulations)

In three-phase systems the oil relative permeability is modeled as a two dimensional function.

If the `SOF3` table is used, $K_{row}$ and $K_{rog}$ are entered as functions of oil saturation. These are the oil relative permeabilities at zero gas saturation and at connate water saturation respectively. These are combined to obtain the three-phase oil relative permeability using a default method, or Stone’s method I or II. The keywords are `STONE1`, `STONE1EX` and `STONE2`.

Other methods are also available to describe the three-phase oil relative permeability that can also be applied to gas and water relative permeabilities:

- The three phase IKU relative permeability is available with the `IKU3P` keyword.
- The three phase ODD3P relative permeability is available with the `ODD3P` keyword.
- Baker’s methods can be used to describe any (or all) of the phase relative permeabilities as two dimensional functions. The keywords are `BAKER1` or `BAKER2`. 

The IKU, ODD3P and Baker methods require the `SOF3`, `SGF3` and `SWF3` keywords to create the two dimensional data.

The relative permeabilities can also be specified as two dimensional functions, avoiding the need to use a special three-phase model within the simulator, using one (or all) of the keywords `SOF32D`, `SGF32D` and `SWF32D`.

**Stability issues**

The following applies in ECLIPSE 300.

If IMPES or AIM Solution methods are used, large capillary pressures may cause instability. This is limited by imposing a maximum rate at which the capillary pressure may change, see keyword `DPCDT`.

**Saturation function keyword family (i)**

The set of saturation function keywords is summarized below:

- `SWOF` If both water and oil are present
- `SGOF` or `SLGOF` If both gas and oil are present

**Saturation function keyword family (ii)**

The keywords are:

- `SOF3` For oil in a three-phase run
- `SOF2` For oil, if present, in a two-phase run
- `SGFN` For gas, if present
- `SWFN` For water, if present
- `SGWFN` For gas-water runs only

**Two dimensional Saturation functions**

Three-phase saturation functions can be described by two-dimensional tables. The keywords are:

- `SOF32D` For oil
- `SWF32D` For gas
- `SGF32D` For water
- `PCG32D` Capillary pressure
- `PCW32D` Capillary pressure

**Two dimensional interpolation methods**

Various interpolation methods can be used to create two dimensional saturation functions. By default gas and water relative permeabilities are one dimensional, and oil relative permeability is two dimensional. Keywords can be used to change the default interpolation method for oil relative permeability or to make the gas and water relative permeability two dimensional. The keywords are:

<table>
<thead>
<tr>
<th>Simulator</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>STONE1</td>
<td></td>
<td>Stone model 1 (modified) for three phase oil permeability.</td>
</tr>
<tr>
<td>Simulator</td>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>---------</td>
<td>-------------</td>
</tr>
<tr>
<td></td>
<td>STONE1EX</td>
<td>Exponent term for Stone model 1.</td>
</tr>
<tr>
<td></td>
<td>STONE2</td>
<td>Stone model 2 (modified) for three phase oil permeability.</td>
</tr>
<tr>
<td>E300 only</td>
<td>BAKER1</td>
<td>Baker interpolation method 1 for three phase permeability. Applies to any specified phase.</td>
</tr>
<tr>
<td>E300 only</td>
<td>BAKER2</td>
<td>Baker interpolation method 2 for three phase permeability. Applies to any specified phase.</td>
</tr>
<tr>
<td>E300 only</td>
<td>IKU3P</td>
<td>IKU interpolation method. Applies to all phases.</td>
</tr>
<tr>
<td>E300 only</td>
<td>ODD3P</td>
<td>ODD3P interpolation method. Applies to all phases.</td>
</tr>
<tr>
<td></td>
<td>SOP3</td>
<td>For oil in three-phase models.</td>
</tr>
<tr>
<td>E300 only</td>
<td>SGF3</td>
<td>For gas in three-phase models.</td>
</tr>
<tr>
<td>E300 only</td>
<td>SWF3</td>
<td>For water in three-phase models.</td>
</tr>
</tbody>
</table>

**Hysteresis**

If hysteresis is being used, both imbibition and drainage curves are entered with the appropriate saturation functions above. The region keyword IMBNUM is then used to assign imbibition table numbers to cells. The SATOPTS keyword should be used to enable the hysteresis treatment. More details of the hysteresis models can be found under "Hysteresis" in the ECLIPSE Technical Description.

If, in addition to conventional hysteresis, WAG hysteresis has been specified via the WAGHYSTR keyword, the additional region keywords WH2NUM and WH3NUM may be optionally used to specify the alternative two-phase and three-phase water relative permeabilities used by the WAG hysteresis model. If these keywords are not specified, this information will be taken from the tables associated with the SATNUM and IMBNUM region keywords. The WAG Hysteresis Model is described in more detail in “Hysteresis in WAG Floods” in the ECLIPSE Technical Description.

**End point scaling**

In some cases the relative permeability and capillary pressure tables vary from cell to cell. Rather than enter large numbers of tables it is possible to scale fixed points on a table to new cell-dependent positions. This may be done horizontally - along the saturation axis - or vertically - rescaling the values of $K_r$ and $P_c$. To use end point scaling, the ENDScale keyword should be included in the RUNSPEC section.

See "Saturation Table Scaling" in the ECLIPSE Technical Description.

**Pressure tables**

Functions of pressure (ROCK, PVDO, PVTQ, PVTW, PVTG) are entered in a form similar to that of the saturation functions. When multiple tables are entered, these may be associated with cells using the PVTONUM keyword. In addition, DENSITY or GRAVITY keyword data is entered for each pressure table region. An alternative for rock properties is to use the ROCKNUM keyword to specify rock table number for each cell.
If an equation of state is being used, PVTO and PVTG tables are not required, as the fluid Z-factors and thus densities can be obtained directly. Viscosities are obtained from the Lorentz, Bray and Clark [Ref. 70] correlation, and only the water surface density is used from the DENSITY keyword. If a KVTABLE table is entered and the KVALUES keyword used in the RUNSPEC section, this is used to define liquid and vapor compositions, but at present an equation of state is still required to define phase densities. (The KVTABLE may also be used in the surface separators.)

**Rock pressure tables**

The following methods exist for specifying the change in rock pore volume and transmissibility as a function of pressure.

- The ROCK keyword sets a reference pressure and compressibility.
- The ROCKTAB keyword enables a table of pore volume ratios and transmissibility multipliers to be input against pressure. The pore volume increases with increasing pressure as the rock material compresses. Hysteretic rock compaction tables may also be entered using keyword ROCKTABH, when the pore space does not recover during reflation. By default, ECLIPSE tabulates the rock compaction multipliers against fluid pressure unless the OVERBURD keyword is used to specify the overburden pressure. In this case, ROCKTAB and ROCKTABH data is tabulated against:
  - effective fluid pressure \( P_{\text{eff}} \) where \( P_{\text{eff}} = P_{\text{fluid}} - \text{Overburden} \)
  - or against:
  - effective stress, where Effective Stress = \( -P_{\text{eff}} \)

The tabulation used depends on the options chosen in the ROCKOPTS keyword.

- In the Coal Bed Methane option, the ROCKPAMA keyword can be used to specify parameters which are used in the Palmer-Mansoori rock model to calculate pore volume and transmissibility multipliers.
- The ROCKTRMX, ROCKTRMY and ROCKTRMZ keywords can be used to define transmissibility as an exponential function of porosity.
- In Dual Porosity runs, keywords ROCKSIG or ROCKTHSG may be used to specify pressure tables of the sigma multiplier for matrix-fracture coupling.

See "Rock Compaction" in the ECLIPSE Technical Description.

**Water pressure tables**

The water compressibility and viscosity may be specified with the PVTW keyword. This might take the form:

```
PVTW
---Pref Bref Cw Vw (dVw/dp)/Vw
4000 1.001 0.000003 0.52 0 /
```

This specifies a reference pressure, the water formation volume factor at this pressure, the compressibility from this pressure and the water viscosity. The last value is a viscosibility, similar to compressibility, but for viscosity. In many cases this is zero. A quadratic function is used for extrapolation.

As an alternative to PVTW, the WATERTAB keyword may be used to enter a general variation of water properties with pressure.
When using the Brine option, the water PVT properties must be entered with the `PVTWSALT` keyword.

### Surface densities

The three possible surface densities are set using the `DENSITY` keyword. This is required when formation volumes are used, in order to define reservoir densities. For compositional runs, the equation of state defines the hydrocarbon reservoir densities, and only the water value is required. `DENSITY` always contains three items, even if water is not present in the run.

```
DENSITY  deno  denw  deng
  45.0000  63.0200  .07020 /
```

If a run has more than one pressure table region set with `TABDIMS` in the `RUNSPEC` section, these must be entered in sequence, each terminated by a slash character (/), for both `DENSITY` and the pressure table data such as `ROCK` and `PVTW`. As an alternative to `DENSITY`, the `GRAVITY` keyword enables the same data to be entered in specific gravity form.

When using the Brine option, the water density can also be entered with the `BDENSITY` keyword.

### Oil and gas pressure tables (black oil)

These are only relevant to black oil mode runs, as the hydrocarbon densities are obtained from the equation of state in compositional mode.

The PVT properties of live oil can be entered in either of two formats, with either:

- `PVCO` and `PMAX`, or
- `PVTO`.

The first format assumes that under saturated oil of a particular $R_s$ value has a compressibility that is independent of the pressure, and that the viscosity of under saturated oil has a pressure-independent derivative. Pressure tables are linearly interpolated, and extrapolated, if required, at constant slope. The second format is more complex, but allows greater flexibility in defining the properties of under saturated oil. It specifies the properties of oil with dissolved gas may be entered in a two dimensional table form.

Similarly, the PVT properties of dead oil can be entered in either of two formats, with keywords `PVCDO` or `PVDO`. Keyword `PVCDO` assumes that dead oil has a constant compressibility and its viscosity has a pressure-independent derivative. Keyword `PVDO` is more flexible, allowing the formation volume factor and the viscosity to be input as a table versus pressure.

For gas, `PVDG` tabulates the gas formation volume factor and viscosity as a function of pressure.

The `Z-factors for dry gas can be entered directly using the `PVZG` keyword.

Finally, the properties of gas with vaporized oil (condensates) may be entered in using the `PVTG` keyword.

Oil and gas pressure tables can be prepared with the `PVT` program.

### Essential keywords - black oil case

The data must contain: surface densities of the fluids, data to calculate the reservoir fluid densities and viscosities, the rock compressibility, and relative permeabilities and capillary pressures as a function of saturation.
The minimum set of keywords required in the PROPS section are listed below.

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keywords for surface gravities/densities</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>E100 only</td>
<td>ROCK</td>
<td>For rock compressibility (unless using the Rock Compaction option)</td>
</tr>
<tr>
<td></td>
<td>PVTO or (PVCO and PMAX)</td>
<td>If live oil is present</td>
</tr>
<tr>
<td>E100 only</td>
<td>PVDO or PVDCO</td>
<td>If dead oil is present</td>
</tr>
<tr>
<td>E100 only</td>
<td>RSCONST or RSCONSTT</td>
<td>If dead oil is used to model oil having a constant dissolved gas concentration</td>
</tr>
<tr>
<td></td>
<td>PVTG</td>
<td>If wet gas is present</td>
</tr>
<tr>
<td></td>
<td>PVDG or PVZG</td>
<td>If dry gas is present</td>
</tr>
<tr>
<td></td>
<td>RVCONST or RVCONSTT</td>
<td>If dry gas is used to model gas having a constant vaporized oil concentration</td>
</tr>
<tr>
<td>E100 only</td>
<td>PVTW</td>
<td>If water is present (but use PVTWSALT instead with the Brine option).</td>
</tr>
</tbody>
</table>

Some keywords are mutually exclusive and cannot be used together. Examples of forbidden combinations are:

- **PVTO** and **RSCONST (T)**. The constant Rs facility is only for dead oil.
- **PVTG** and **RVCONST (T)**. The constant Rv facility is only for dry gas.
- **SWOF** and **SGFN**. The two saturation keyword families cannot be mixed.

The table below shows the choice of keywords that may be used in various examples of phase combinations. **PVZG** may be used in place of **PVDG** for dry gas properties.

<table>
<thead>
<tr>
<th>Phase Combinations</th>
<th>Keywords</th>
</tr>
</thead>
</table>
| Oil and Dissolved Gas, Wet Gas and Vaporized Oil Water | either: PVCO, PMAX, PVTG, PVTW
|                                                       | or: PVTO, PVTG, PVTW
|                                                       | either: SGOF or SLGOF, SWOF
|                                                       | or: SOF3 or SOF32D, SGFN, SWFN                                          |
| Oil and Dissolved Gas, Dry Gas, Water                 | either: PVCO, PMAX, PVDG, PVTW                                          |
|                                                       | or: PVTO, PVDG, PVTW                                                    |
|                                                       | either: SGOF or SLGOF, SWOF                                              |
|                                                       | or: SOF3 or SOF32D, SGFN, SWFN                                          |
| Oil and Dissolved Gas, Wet Gas and Vaporized Oil      | either: PVCO, PMAX, PVTG                                                |
|                                                       | or: PVTO, PVTG                                                         |
|                                                       | either: SGOF or SLGOF                                                   |
|                                                       | or: SOF2, SGFN                                                         |
### Table 2.7: PROPS keywords to be used with the different phase combinations in a black oil run

<table>
<thead>
<tr>
<th>Phase Combinations</th>
<th>Keywords</th>
</tr>
</thead>
</table>
| Undersaturated Oil and Constant Dissolved Gas, Water | either: $\text{PVDO}$, $\text{RSCONST(T)}$, $\text{PVTW}$  
or: $\text{PVCDG}$, $\text{RSCONST(T)}$, $\text{PVTW}$  
either: $\text{SWOF}$  
or: $\text{SOF2}$, $\text{SWFN}$ |
| Dry Gas | $\text{PVDG}$ or $\text{PVZG}$ |
| Dry Gas and Constant Vaporized Oil, Water | $\text{RVCONST(T)}$, $\text{PVDG}$, $\text{PVTW}$  
either: $\text{SGFN}$, $\text{SWFN}$  
or: $\text{SGWFN}$ |

#### Standard conditions (ECLIPSE 300)

Surface gas volumes are reported as ideal gas volumes at standard conditions - by default 1 atmosphere and 15.56 °C (60 °F). Oil volumes are reported at final stage separator conditions. The $\text{STCOND}$ keyword can be used to override the default conditions.

#### Compositional Equation of State data (ECLIPSE 300)

In compositional mode, the equation of state may be chosen using the $\text{EOS}$ keyword. The single item should start with P, R, S or Z, for Peng-Robinson, Redlich-Kwong, Soave-Redlich-Kwong and Zudkevitch-Joffe equations respectively. A small correction to the Peng-Robinson is available using the $\text{PRCORR}$ keyword. The equations of state are discussed in "Equations of State" in the ECLIPSE Technical Description.

The PROPS section contains keywords to parameterize the equation of state. The following contain $N_c$ items, where $N_c$ is the number of hydrocarbon components in the run, specified with $\text{COMPS}$ in the RUNSPEC section:

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\text{C NAMES}$</td>
<td>Component names</td>
</tr>
<tr>
<td>$\text{TCRIT}$</td>
<td>Critical temperatures</td>
</tr>
<tr>
<td>$\text{PCRIT}$</td>
<td>Critical pressures</td>
</tr>
<tr>
<td>$\text{VCRIT}$</td>
<td>Critical volumes</td>
</tr>
<tr>
<td>$\text{ZCRIT}$</td>
<td>Critical Z-factors</td>
</tr>
<tr>
<td>$\text{VCRITVIS}$</td>
<td>Critical volumes (for viscosities only)</td>
</tr>
<tr>
<td>$\text{MW}$</td>
<td>Molecular weights</td>
</tr>
<tr>
<td>$\text{ACF}$</td>
<td>Acentric factors</td>
</tr>
<tr>
<td>$\text{TBOIL}$</td>
<td>Boiling points (for Zudkevitch Joffe)</td>
</tr>
<tr>
<td>$\text{DREF}$</td>
<td>Reference densities (for Zudkevitch Joffe)</td>
</tr>
<tr>
<td>$\text{GREF}$</td>
<td>Reference gravities (for Zudkevitch Joffe)</td>
</tr>
</tbody>
</table>
### ECLIPSE 300 only keyword

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TREF</td>
<td>Reference temperatures (for Zudkevitch Joffe)</td>
</tr>
<tr>
<td>OMEGAA</td>
<td>$\Omega_{a0}$ values for reservoir</td>
</tr>
<tr>
<td>OMEGAB</td>
<td>$\Omega_{b0}$ values for reservoir</td>
</tr>
<tr>
<td>PARACHOR</td>
<td>Parachor values for surface tension evaluation</td>
</tr>
<tr>
<td>SSHIFT</td>
<td>Equation of state volume shifts for reservoir</td>
</tr>
</tbody>
</table>

Note that **either** Z-factors or critical volumes should be entered, as these are related by the relationship $V_c = Z_cRT/P_c$. The ZCRITVIS or VCRITVIS keywords need only be entered if different critical volumes are required for the Lorentz-Bray-Clark viscosity correlation. If an alternate method to this default viscosity calculation is required, then the Pedersen et al. viscosity calculation can be used by including the PEDERSEN keyword in this PROPS section.

The coefficients of the Lorentz-Bray-Clark correlation may be modified using the LBCCOEFL or LBCCOEFR keywords. A modified form of the Lorentz-Bray-Clark correlation can be invoked by using keyword VCOMPACT. For the Pedersen et al. viscosity correlation, the coefficients used in the mixture molecular weight calculation can be modified by multiplicative tuning parameters which may be specified using either the PEDTUNE or the PEDTUNER keywords.

In addition to the above properties, binary interaction coefficients may be entered with the BIC keyword. $N_c(N_c-1)/2$ values are entered, the lower triangular part of the interaction coefficient matrix. For example, for a five component system:

```
<table>
<thead>
<tr>
<th></th>
<th>BIC</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.01</td>
</tr>
<tr>
<td>0.01</td>
<td>0.005</td>
</tr>
<tr>
<td>0.02</td>
<td>0.0</td>
</tr>
<tr>
<td>0.001</td>
<td>0.0</td>
</tr>
</tbody>
</table>
```

To facilitate the matching of PVT data, the simulator allows the use of two sets of EoS data, one for the reservoir and one for the surface conditions. It is possible to use different equation of state parameters for surface conditions using keywords such as ACFS, OMEGAA, OMEGABS, BICS and SSHIFTS. Furthermore, the EoS at the surface (say ZJ) may be different from the EoS used for the reservoir (say PR). A different surface EoS may be specified using EOSS.

For compositional runs it is also required to enter the reservoir temperature using RTEMP, used to define reduced temperatures for the equation of state. Note that this is read in degrees F or degrees C. An alternative is the TEMPVD keyword, which allows for a temperature variation with respect to depth in the reservoir, and alters the equation of state coefficients for each cell accordingly. If the equation of state data is prepared using the PVT $i$ pre-processor program, the file to be included in the ECLIPSE 300 dataset includes the NCOMPS keyword. When data is prepared in this way the NCOMPS keyword is read and checked, to confirm that the number of components assumed by pre-processor and simulator agree. NCOMPS does not need to be entered.

### K-value tables (ECLIPSE 300)

If K-value tables are to be used in the reservoir (KVALUES in RUNSPEC) or for the separators, these may be entered in the PROPS section using the KVTABLE keyword as functions of pressure. $N_c$ K-values are entered at each pressure value. K-value dependence on composition can be model by interpolating between
a set of K-value tables depending on the mole fraction of a nominated component. The K-value tables are
supplied using the KVTABCn keywords, the interpolation points via the KVMF keyword and the component
specified with the KVCOMP keyword.

Generic table reading routines are used to read many ECLIPSE 300 tables, including KVTABLE, KVTABCn
and KVABTn. If values in a table are defaulted (*), the table values are constructed using linear
interpolation and constant extrapolation. For instance, below are four examples which demonstrate the
values used internally when defaults are used in the KVTABLE table (with one component).

For Example 1, linear interpolation will be used to define the K-values for the defaulted pressure values of
200 and 300:

```
KVTABLE
  --p_oil  K-value
  100.0   1
  200.0   *    (internal value used = 2)
  300.0   *    (internal value used = 3)
  400.0   4
/```

For Example 2, constant extrapolation will be used to define the K-values for the defaulted pressure values
of 100 and 400:

```
KVTABLE
  --p_oil  K-value
  100.0   *    (internal value used = 2)
  200.0   2
  300.0   3
  400.0   *    (internal value used = 3)
/```

For Example 3, constant extrapolation will be used to define the K-values for the defaulted pressure values
of 200, 300 and 400:

```
KVTABLE
  --p_oil  K-value
  100.0   1
  200.0   *    (internal value used = 1)
  300.0   *    (internal value used = 1)
  400.0   *    (internal value used = 1)
/```

For Example 4, constant extrapolation will be used to define the K-values for the defaulted pressure values
of 100, 300 and 400:

```
KVTABLE
  --p_oil  K-value
  100.0   *    (internal value used = 2)
  200.0   2
  300.0   *    (internal value used = 2)
  400.0   *    (internal value used = 2)
/```

As an alternative to entering K-values in tabular form, the KVGEN keyword may be used to request that
they be generated internally from a given sample. These may be used either to control the interphase
equilibrium over the field (the KVALUES keyword in the RUNSPEC section) or in separator calculations
(the SEPCOND keyword in the SCHEDULE section).
Diffusion coefficients

In the Diffusion option is being used (DIFFUSE in the RUNSPEC section) then normal or activity corrected diffusion coefficients may be specified for each component.

See "Diffusion" in the ECLIPSE Technical Description.

Composition with respect to pressure tables (ECLIPSE 300)

The normal method of defining the initial state of a simulation is by equilibration. However, it is sometimes convenient to specify the initial pressures and saturations explicitly. Great care must be taken to ensure that this the initial state specified is stable and physical. The XMFP and YMFVP keywords allow the initial oil and gas compositions, expressed as mole fractions, to be entered as functions of pressure, so that equilibrium compositions may be obtained over a range of pressures. These tables would normally be set up by the PVT i program, using a constant volume depletion or differential liberation experiment to define the composition variation with pressure.

PROPS section keywords

A list of all keywords which may occur in the PROPS section grouped by function is given below, together with a brief description of their use. This is followed by an alphabetic list. A more detailed description may be found by looking up the keyword in the manual.

Alkaline option

Common keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required if Surfactant option is active.</td>
<td>ALSURFST</td>
<td>Oil/water surface tension multipliers.</td>
</tr>
</tbody>
</table>

ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keywords</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required if Surfactant option is active.</td>
<td>ALSURPAD</td>
<td>Surfactant adsorption multipliers.</td>
</tr>
<tr>
<td>Required if Polymer option is active.</td>
<td>ALPOLADS</td>
<td>Polymer adsorption multipliers.</td>
</tr>
<tr>
<td></td>
<td>ALKADS</td>
<td>Alkaline adsorption functions (tabulated).</td>
</tr>
<tr>
<td></td>
<td>ADSORP</td>
<td>Alkaline adsorption functions (analytical).</td>
</tr>
<tr>
<td>Required if ALKADS or ADSORP is specified.</td>
<td>ALKROCK</td>
<td>Alkaline-rock properties.</td>
</tr>
</tbody>
</table>
## API tracking (ECLIPSE 100)

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>API tracking only.</td>
<td>APIGROUP</td>
<td>Allows different groups of oil PVT tables to be used in different PVNUM regions.</td>
</tr>
</tbody>
</table>

## Black Oil fluid PVT property data

### Common keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DENSITY or GRAVITY required.</td>
<td>DENSITY</td>
<td>Stock tank fluid densities.</td>
</tr>
<tr>
<td>DENSITY or GRAVITY</td>
<td>GRAVITY</td>
<td>Stock tank fluid gravities. required.</td>
</tr>
<tr>
<td>Required when PVCO is used, otherwise optional.</td>
<td>PMAX</td>
<td>Maximum pressure expected in the simulation run.</td>
</tr>
<tr>
<td>PVCO or PVTO required for live oil systems.</td>
<td>PVCO</td>
<td>FVF and viscosity of live oil as functions of bubble point pressure.</td>
</tr>
<tr>
<td>PVCO or PVTO required for live oil systems.</td>
<td>PVTO</td>
<td>FVF and viscosity of live oil as functions of pressure and Rs.</td>
</tr>
<tr>
<td>For systems with wet gas. This keyword is required.</td>
<td>PVTG</td>
<td>FVF and viscosity of gas as functions of pressure and Rv.</td>
</tr>
<tr>
<td>PVDG or PVZG required for dry gas systems</td>
<td>PVDG</td>
<td>FVF and viscosity of dry gas as functions of pressure.</td>
</tr>
<tr>
<td>For systems with water, but not with the Brine option. This keyword is required.</td>
<td>PVTW</td>
<td>FVF, compressibility and viscosity of water.</td>
</tr>
</tbody>
</table>

### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PVCDO or PVDO required for dead or under saturated oil.</td>
<td>PVCDO</td>
<td>FVF and viscosity of dead oil with constant compressibility.</td>
</tr>
<tr>
<td>PVCDO or PVDO required for dead or under saturated oil.</td>
<td>PVDO</td>
<td>FVF and viscosity of dead oil as functions of pressure.</td>
</tr>
<tr>
<td>Dead oil only.</td>
<td>RSCONST</td>
<td>Sets a constant Rs for dead oil over the whole field.</td>
</tr>
<tr>
<td>Dead oil only.</td>
<td>RSCONSTT</td>
<td>Sets constant Rs for dead oil, a different value in each PVT region.</td>
</tr>
<tr>
<td>PVDG or PVZG required for dry gas systems.</td>
<td>PVZG</td>
<td>Z-factor and viscosity of dry gas as functions of pressure.</td>
</tr>
<tr>
<td>Dry gas only.</td>
<td>RVCONST</td>
<td>Sets a constant Rv for dry gas over the whole field.</td>
</tr>
<tr>
<td>Dry gas only.</td>
<td>RVCONSTT</td>
<td>Sets constant Rv for dry gas, a different value in each PVT region.</td>
</tr>
</tbody>
</table>
ECLIPSE 300 keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required</td>
<td>WATERTAB</td>
<td>Water pressure tables. Alternative to PVTW for systems with water.</td>
</tr>
</tbody>
</table>

Brine option

Common keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required</td>
<td>PVTWSALT</td>
<td>FVF, compressibility and viscosity of water as functions of salt concentration.</td>
</tr>
<tr>
<td></td>
<td>BDENSITY</td>
<td>Enables the surface density of water to vary with the salt concentration.</td>
</tr>
</tbody>
</table>

ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Brine option with dispersion only.</td>
<td>DSPDEINT</td>
<td>When using dispersion on a brine tracer using this keyword the second item of the dispersion table (concentration of the tracer) becomes water density. The interpolation of the dispersion coefficient will then be performed on velocity and water density.</td>
</tr>
<tr>
<td></td>
<td>IONXROCK</td>
<td>Rock ion exchange parameters for cation exchange.</td>
</tr>
<tr>
<td></td>
<td>IONXSURF</td>
<td>Surfactant micelle ion exchange parameters for cation exchange.</td>
</tr>
<tr>
<td></td>
<td>PLYESAL</td>
<td>Coefficients for polymer effective salinity.</td>
</tr>
<tr>
<td></td>
<td>SURFESAL</td>
<td>Coefficients for surfactant effective salinity.</td>
</tr>
</tbody>
</table>

Carbon Dioxide solution in aqueous phase (ECLIPSE 300)

This option is for ECLIPSE 300 only. The keywords are:

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CO2SOL and GASSOL options.</td>
<td>RSWVD</td>
<td>Aqueous concentration of dissolved gases versus depth.</td>
</tr>
<tr>
<td></td>
<td>DENAQA</td>
<td>This keyword enables the default Ezrokhi coefficients for the aqua density calculation to be overwritten.</td>
</tr>
<tr>
<td></td>
<td>VISCAQA</td>
<td>This keyword enables the default Ezrokhi coefficients for the aqua viscosity calculation to be overwritten.</td>
</tr>
</tbody>
</table>

CO2SOL only keywords

- **SOLUBILI** Specify properties of water-CO$_2$ system.
- **SOLUBILS** Specify properties of water-CO$_2$ system with salt dependencies.
- **SOLUBILT** Specify properties of water-CO$_2$ system with temperature dependencies.
- **SOLUCO2H** Specify properties of water-CO$_2$ system with H$_2$S dependencies.
• **SOLUH2SC** Specify properties of water-H$_2$S system with CO$_2$ dependencies.

**GASSOL only keywords**

• **SOLUAQA** Specify properties of a water-multi component gas system.

**CO2STORE only keywords**

• **ACTCO2S** Activity coefficient model for CO$_2$

**Carter-Tracy aquifer data**

• **AQUTAB** Influence functions used by Carter-Tracy aquifers.

**Chemical reaction keywords (ECLIPSE 300)**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal</td>
<td>EQLDKVCR</td>
<td>K-values for equilibrium deviation reactions</td>
</tr>
<tr>
<td></td>
<td>EQLDREAC</td>
<td>Equilibrium deviation reaction rates</td>
</tr>
<tr>
<td></td>
<td>REACACT</td>
<td>Reaction activation molar energy</td>
</tr>
<tr>
<td></td>
<td>REACENTH</td>
<td>Reaction molar enthalpy</td>
</tr>
<tr>
<td></td>
<td>REACCORD</td>
<td>Reaction rate component order</td>
</tr>
<tr>
<td></td>
<td>REACPORD</td>
<td>Reaction rate porosity order</td>
</tr>
<tr>
<td></td>
<td>REACRATE</td>
<td>Reaction rate constant</td>
</tr>
<tr>
<td>Solid</td>
<td>REACSMAX</td>
<td>Reaction rate solid saturation limit.</td>
</tr>
<tr>
<td>Solid</td>
<td>REACSORD</td>
<td>Reaction rate solid order.</td>
</tr>
<tr>
<td></td>
<td>REACTRAC</td>
<td>Tracer dependent reaction rate terms</td>
</tr>
<tr>
<td></td>
<td>STOPROD</td>
<td>Stoichiometric coefficients of products</td>
</tr>
<tr>
<td></td>
<td>STOREAC</td>
<td>Stoichiometric coefficients of reactants</td>
</tr>
</tbody>
</table>

**Coal bed methane (ECLIPSE 100)**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Either COALADS or COALPP should be specified.</td>
<td>COALADS</td>
<td>Gas/solvent relative adsorption data.</td>
</tr>
<tr>
<td>Either COALADS or COALPP should be specified.</td>
<td>COALPP</td>
<td>Gas/solvent partial pressure data.</td>
</tr>
<tr>
<td></td>
<td>DIFFCOAL</td>
<td>Gas diffusion data.</td>
</tr>
<tr>
<td></td>
<td>LANGMPL</td>
<td>Pressure scaling factor for the adsorption capacity for each grid cell.</td>
</tr>
</tbody>
</table>
### Coal bed methane (ECLIPSE 300)

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DIFFCBM</td>
<td>Diffusion coefficients.</td>
</tr>
<tr>
<td>LANGMEXT</td>
<td>Extended Langmuir isotherm data for each component.</td>
</tr>
<tr>
<td>LANGMPL</td>
<td>Pressure scaling factor for the adsorption capacity for each grid cell.</td>
</tr>
<tr>
<td>LANGMUIR</td>
<td>Langmuir isotherm table data for each component.</td>
</tr>
<tr>
<td>LANGMULC</td>
<td>Langmuir multipliers on a cell by cell basis for each component.</td>
</tr>
<tr>
<td>LANGMULT</td>
<td>Langmuir multipliers on a cell by cell basis.</td>
</tr>
<tr>
<td>RESORB</td>
<td>Re-adsorption factor.</td>
</tr>
<tr>
<td>ROCKPAMA</td>
<td>Palmer-Mansoori rock model data.</td>
</tr>
<tr>
<td>ROCKPAME</td>
<td>Composition dependent Palmer-Mansoori alike rock model data.</td>
</tr>
<tr>
<td>TPAMEPS</td>
<td>Specify the strain term for each component versus gas concentration to be used by ROCKPAMA.</td>
</tr>
</tbody>
</table>

### Compositional EoS and PVT property data (ECLIPSE 300)

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>ACF</td>
<td>Acentric factors. Required.</td>
</tr>
<tr>
<td>Defaults to ACF if not specified</td>
<td>ACFS</td>
<td>Acentric factors for surface EoS</td>
</tr>
<tr>
<td>Required</td>
<td>BIC</td>
<td>Binary interaction coefficients for EoS.</td>
</tr>
<tr>
<td>Defaults to BIC if not specified</td>
<td>BICS</td>
<td>Binary interaction coefficients for surface EoS</td>
</tr>
<tr>
<td>Required</td>
<td>CALVAL / CALVALR</td>
<td>Component calorific values and component calorific values for multiple equation of state regions.</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 300 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>----------------------------------------------------------------------------</td>
<td>----------------------------</td>
<td>-----------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>For calorific value rate control of groups and wells (and the GASWAT option. Required)</td>
<td>CNUM</td>
<td>Component names.</td>
</tr>
<tr>
<td>DENSITY or GRAVITY is required only for water, if present.</td>
<td>DENSITY</td>
<td>Stock tank fluid densities.</td>
</tr>
<tr>
<td>DENSITY or GRAVITY is required only for water, if present.</td>
<td>GRAVITY</td>
<td>Stock tank fluid gravities.</td>
</tr>
<tr>
<td>Required for Zudkevitch Joffe only.</td>
<td>DREF</td>
<td>Reference densities (for Zudkevitch Joffe EoS).</td>
</tr>
<tr>
<td>Required (but assumes default values if omitted).</td>
<td>GREF</td>
<td>Reference gravities (for Zudkevitch Joffe).</td>
</tr>
<tr>
<td>Used only if the default coefficients in the Lorentz-Bray-Clark viscosity correlation are changed.</td>
<td>LBCCOEF</td>
<td>Set non-default LBC coefficients.</td>
</tr>
<tr>
<td>Used only if the default coefficients in the Lorentz-Bray-Clark viscosity correlation are changed.</td>
<td>LBCCOEFR</td>
<td>Set non-default LBC coefficients for multiple equation of state regions.</td>
</tr>
<tr>
<td>Defaults to MW if not specified. Required</td>
<td>MWS</td>
<td>Molecular weights.</td>
</tr>
<tr>
<td>Defaults to OMEGAA if not specified. Required</td>
<td>OMEGAA</td>
<td>$\Omega_a$ values for reservoir.</td>
</tr>
<tr>
<td>Defaults to OMEGAA if not specified. Required</td>
<td>OMEGAA</td>
<td>$\Omega_a$ values for surface EoS.</td>
</tr>
<tr>
<td>Defaults to OMEGAB if not specified. Required</td>
<td>OMEGAB</td>
<td>$\Omega_b$ values for reservoir. Assumes default values if omitted.</td>
</tr>
<tr>
<td>Defaults to OMEGAB if not specified. Required</td>
<td>OMEGABS</td>
<td>$\Omega_b$ values for surface EoS.</td>
</tr>
<tr>
<td>Defaults to PCRIT if not specified. Required</td>
<td>PCRIT</td>
<td>Critical pressures.</td>
</tr>
<tr>
<td>Defaults to PCRIT if not specified. Required</td>
<td>PCRTS</td>
<td>Critical pressures for surface EoS.</td>
</tr>
<tr>
<td>Required for Pedersen et al. viscosity calculation only.</td>
<td>PEDTUNE</td>
<td>Provides a single set of multiplicative tuning parameters to modify the mixture molecular weight calculation used in the Pedersen et al. viscosity calculation.</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 300 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-------------------------------------------------</td>
<td>--------------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Optional for Pedersen et al. viscosity calculation only.</td>
<td>PEDTUNER</td>
<td>Provides multiplicative tuning parameters to modify the mixture molecular weight calculation used in the Pedersen et al. viscosity calculation specified separately for each equation of state region.</td>
</tr>
<tr>
<td>Optional for Pedersen et al. viscosity calculation only.</td>
<td>PREF</td>
<td>Reference pressures for the Thermal option.</td>
</tr>
<tr>
<td>Optional for Pedersen et al. viscosity calculation only.</td>
<td>PREFS</td>
<td>Reference pressures for surface EoS in the Thermal option.</td>
</tr>
<tr>
<td>Optional when using PR EoS</td>
<td>RTEMP</td>
<td>Request modified Peng-Robinson EoS.</td>
</tr>
<tr>
<td>RTEMP or TEMPVD is required</td>
<td>SSHIFT</td>
<td>Equation of state volume shifts for reservoir. Defaults to 0 if not specified.</td>
</tr>
<tr>
<td>Defaults to SSHIFT if not specified.</td>
<td>SSHIFTS</td>
<td>Equation of state volume shifts for surface Eos.</td>
</tr>
<tr>
<td></td>
<td>STCOND</td>
<td>Specify standard conditions. Defaults if not specified.</td>
</tr>
<tr>
<td></td>
<td>TBOIL</td>
<td>Boiling points (for Zudkevitch Joffe EoS).</td>
</tr>
<tr>
<td>Required for Zudkevitch Joffe only</td>
<td>TBOILS</td>
<td>Boiling points for surface (for Zudkevitch Joffe EoS).</td>
</tr>
<tr>
<td>RTEMP or TEMPVD is required</td>
<td>TCRIT</td>
<td>Critical temperatures.</td>
</tr>
<tr>
<td>Required for Zudkevitch Joffe only</td>
<td>TCRITS</td>
<td>Critical temperatures for surface EoS.</td>
</tr>
<tr>
<td></td>
<td>TEMPVD</td>
<td>Specify constant temperature versus depth.</td>
</tr>
<tr>
<td>Required for Zudkevitch Joffe only</td>
<td>TREF</td>
<td>Reference temperatures (for Zudkevitch Joffe).</td>
</tr>
<tr>
<td>Defaults to VCRIT if not specified.</td>
<td>VCRITVIS</td>
<td>Critical volumes (for viscosities only).</td>
</tr>
<tr>
<td>Defaults to ZCRIT if not specified.</td>
<td>ZCRIT</td>
<td>Critical Z-factors.</td>
</tr>
<tr>
<td>Defaults to ZCRIT or if not specified</td>
<td>ZCRITVIS</td>
<td>Critical Z-factors (for viscosities only).</td>
</tr>
</tbody>
</table>

**Dual porosity (ECLIPSE 100)**

- **DPKRMOD** Modifies the oil relative permeability in dual porosity runs. For dual porosity runs only.
- **INTPC** Invokes the dual porosity integrated P_c option For dual porosity runs only.
## End point scaling

### Common keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>ENKRVD</td>
<td>Variation of end point relative permeability values with depth.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>ENPCVD</td>
<td>Variation of maximum capillary pressure values with depth.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>ENPTVD</td>
<td>Variation of end point saturation values with depth.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>EPSDBGS</td>
<td>Writes scaled relative permeability curves to the Debug file, for multiple sets of grid blocks.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>EPSDEBUG</td>
<td>Writes scaled relative permeability curves to the Debug file, for grid blocks in the specified box.</td>
</tr>
<tr>
<td>Saturation table end point scaling with hysteresis only.</td>
<td>IKRG, IKRGR, IKRW, IKRW, IKRO, IKRORG, IKRORW</td>
<td>Imbibition curve end point relative permeability values, equivalent to keywords KRG, KRGR, KRW and KRWR for example.</td>
</tr>
<tr>
<td>Saturation table end-point scaling option and WAG hysteresis option only.</td>
<td>IKRWH</td>
<td>Scales three-phase water relative permeability values at maximum water saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end-point scaling option and WAG hysteresis option only.</td>
<td>IKRWRH</td>
<td>Scales three-phase water relative permeability values at residual oil saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling with hysteresis only.</td>
<td>ISGL, ISGCR, ISGU, ISWL, ISWCR, ISWU, ISGCR, ISOWCR</td>
<td>Imbibition curve end point scaling data, equivalent to keywords including SGL, SGCR and SGU.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>KRG</td>
<td>Scales gas relative permeability values at maximum gas saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>KRGR</td>
<td>Scales gas relative permeability values at residual oil saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>KRO</td>
<td>Scales oil relative permeability values at maximum oil saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>KROG</td>
<td>Scales oil relative permeability values at critical gas saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>KROW</td>
<td>Scales oil relative permeability values at critical water saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>KRW</td>
<td>Scales water relative permeability values at maximum water saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>Restriction</td>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>--------------------------------------------------------------</td>
<td>---------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Saturation table end point scaling and WAG hysteresis option only.</td>
<td>KRWH</td>
<td>Scales two-phase water relative permeability values at maximum water saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>KRWR</td>
<td>Scales water relative permeability values at residual oil saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option and WAG hysteresis option only.</td>
<td>KRWRH</td>
<td>Scales two-phase water relative permeability values at residual oil saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>PCG</td>
<td>Scales the maximum gas capillary pressure, for cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>PCW</td>
<td>Scales the maximum water capillary pressure, for cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>PPCWMAX</td>
<td>Limits the calculated PCW values when the SWATINIT option is used.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>SCALECRS</td>
<td>Requests alternative end point scaling method.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>SGCR</td>
<td>Critical gas saturation for grid cells in the current input box.  [SWATINIT]</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>SGL</td>
<td>Connate gas saturation for grid cells in the current input box.  [SWATINIT]</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>SGU</td>
<td>Maximum gas saturation for grid cells in the current input box. For saturation table end point scaling option only.  [SOGCR] Residual oil-in-gas saturation for grid cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only using the IKU three-phase relative permeability model with IKU3P in the PROPS section</td>
<td>SWGCR</td>
<td>Residual water-in-gas saturation for grid cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>SOWCR</td>
<td>Residual oil-in-water saturation for grid cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>SWATINIT</td>
<td>Initial water saturation, for grid cells in the current input box, to scale water (P_c) curves.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>SWCR</td>
<td>Critical water saturation for grid cells in the current input box.  [SWATINIT]</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>SWL</td>
<td>Connate water saturation for grid cells in the current input box.  [SWATINIT]</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>SWU</td>
<td>Maximum water saturation for grid cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>TOLCRIT</td>
<td>Controls the method of determining the table critical saturations in pathological end point scaling cases.</td>
</tr>
<tr>
<td>Mobile fluid correction cases.</td>
<td>TZONE</td>
<td>Controls the variation of critical end-point with depth.</td>
</tr>
</tbody>
</table>
### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>ENSPCVD</td>
<td>Variation of the scaled connate saturation, for $P_c$ curves only, with depth.</td>
</tr>
<tr>
<td>Saturation table end-point scaling with hysteresis only.</td>
<td>IMKRVD</td>
<td>Variation of end point relative permeability values with depth for the imbibition curve.</td>
</tr>
<tr>
<td>Saturation table end-point scaling with hysteresis only.</td>
<td>IMPCVD</td>
<td>Variation of maximum capillary pressure values with depth for the imbibition curve.</td>
</tr>
<tr>
<td>Saturation table end-point scaling with hysteresis only.</td>
<td>IMPTVD</td>
<td>Variation of end point saturation values with depth for the imbibition curve.</td>
</tr>
<tr>
<td>Saturation table end point scaling with hysteresis only.</td>
<td>IMSPCVD</td>
<td>Variation of the scaled connate saturation, for $P_c$ curves only, with depth for the imbibition curve.</td>
</tr>
<tr>
<td>Saturation table end point scaling with hysteresis only.</td>
<td>IPCG, IPCW</td>
<td>Imbibition curve end point capillary pressure values, equivalent to keywords PCG, PCW.</td>
</tr>
<tr>
<td>Saturation table end point scaling with hysteresis only.</td>
<td>ISGLPC, ISWLPC</td>
<td>Imbibition curve end point scaling data, equivalent to keywords SGLPC, SWLPC.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>NOWARNEP</td>
<td>Suppresses all warning messages resulting from end point consistency checks.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>SCALELIM</td>
<td>Sets saturation scaling limits for each end point scaling versus depth region.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>SGLPC</td>
<td>Connate gas saturation, for $P_c$ curves only, for grid cells in the current input box.</td>
</tr>
<tr>
<td>Saturation table end point scaling option only.</td>
<td>SWLPC</td>
<td>Connate water saturation, for $P_c$ curves only, for grid cells in the current input box.</td>
</tr>
</tbody>
</table>

### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Saturation table end point scaling option only using the IKU three-phase relative permeability model with 1KU3P in the PROPS section.</td>
<td>SGWCR</td>
<td>Residual gas-in-water saturation for grid cells in the current input box.</td>
</tr>
</tbody>
</table>

### Environmental tracers (ECLIPSE 100)

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keywords</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optional, required for partitioned tracers</td>
<td>TRACERKP</td>
<td>Partitioning function $K(P)$ for partitioned tracers.</td>
</tr>
<tr>
<td>Optional, required for partitioned tracers</td>
<td>TRACERKM</td>
<td>Partitioning functions $K(P)$ for generalized, multi-partitioned tracers.</td>
</tr>
<tr>
<td>Optional, required for any adsorbing tracer</td>
<td>TRADS</td>
<td>Tracer adsorption functions (tabulated).</td>
</tr>
</tbody>
</table>

Data File Overview
Restriction | ECLIPSE 100 only keywords | Description
--- | --- | ---
Optional, required for any adsorbing tracer | ADSORP | Tracer adsorption functions (analytical).
Optional, required for any decaying tracer | TRDCY | Tracer decay functions.
Optional, required for any diffusing tracer | TRDIF | Tracer diffusion functions.
Optional, required for any adsorbing tracer | TRROCK | Tracer rock properties.

**Foam Model**

This option is available in both simulators, though some keywords are only available in ECLIPSE 100.

**Common keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Functional model</td>
<td>FOAMFCN</td>
<td>Gas mobility reduction dependence upon capillary number</td>
</tr>
<tr>
<td>Functional model</td>
<td>FOAMFRM</td>
<td>Gas reference mobility reduction factor</td>
</tr>
<tr>
<td>Functional model</td>
<td>FOAMFSC</td>
<td>Gas mobility reduction dependence upon surfactant concentration</td>
</tr>
<tr>
<td>Functional model</td>
<td>FOAMFSO</td>
<td>Gas mobility reduction dependence upon oil saturation</td>
</tr>
<tr>
<td>Functional model</td>
<td>FOAMFST</td>
<td>Gas-water surface tension dependence upon concentration</td>
</tr>
<tr>
<td>Functional model</td>
<td>FOAMFSW</td>
<td>Gas mobility reduction dependence upon water saturation</td>
</tr>
</tbody>
</table>

**ECLIPSE 100 only keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analytical</td>
<td>ADSORP</td>
<td>Foam adsorption functions</td>
</tr>
<tr>
<td>Tabulated</td>
<td>FOAMADS</td>
<td>Foam adsorption functions</td>
</tr>
<tr>
<td>Tabular model</td>
<td>FOAMDCYO</td>
<td>Foam decay as a function of oil saturation</td>
</tr>
<tr>
<td>Tabular model</td>
<td>FOAMDCYW</td>
<td>Foam decay as a function of water saturation</td>
</tr>
<tr>
<td>Tabular model, optional</td>
<td>FOAMMOB</td>
<td>Gas mobility reduction data</td>
</tr>
<tr>
<td>Tabular model</td>
<td>FOAMMOBP</td>
<td>Pressure dependence on foam mobility reduction</td>
</tr>
<tr>
<td>Tabular model</td>
<td>FOAMMOBS</td>
<td>Shear dependence on foam mobility reduction</td>
</tr>
<tr>
<td>Tabular model</td>
<td>FOAMOPTS</td>
<td>Specifies the Foam Model options.</td>
</tr>
<tr>
<td>Tabular model</td>
<td>FOAMROCK</td>
<td>Specifies the foam rock properties.</td>
</tr>
</tbody>
</table>

**GASWAT option (ECLIPSE 300)**

The keywords are:

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>AQCOEF</td>
<td>A-coefficient terms for water component</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 300 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-------------</td>
<td>--------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td></td>
<td>BICAQ1 BICAQ2 BICAQ3</td>
<td>Binary interaction coefficients for aqueous phase</td>
</tr>
<tr>
<td>Required for the GASWAT option</td>
<td>CALVAL, CALVALR</td>
<td>Component calorific values and component calorific values for multiple equation of state regions.</td>
</tr>
<tr>
<td>Required for the GASWAT option</td>
<td>GSF</td>
<td>Gas saturation functions.</td>
</tr>
<tr>
<td>Required for the GASWAT option</td>
<td>WSF</td>
<td>Water saturation functions.</td>
</tr>
<tr>
<td></td>
<td>SALINITR, SALINITY</td>
<td>Reservoir salinity and regional reservoir salinity</td>
</tr>
</tbody>
</table>

### Geomechanics (ECLIPSE 300)

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BIOTCTAB</td>
<td>Defines a multiplier for the Biot constant, this being a two-dimensional tabular function of maximum and minimum principal stress.</td>
</tr>
<tr>
<td>GEOYLD</td>
<td>Defines parameters for the plastic yield function in each geomechanics region.</td>
</tr>
<tr>
<td>PERMSTAB</td>
<td>Defines a permeability multiplier, this being a two-dimensional tabular function of maximum and minimum principal stress. A table is entered for each geomechanics region.</td>
</tr>
<tr>
<td>SCDATABG</td>
<td>Defines a scale damage table to be used together with the proppant flowback feature, keyword PROPFLOW.</td>
</tr>
</tbody>
</table>

### GI pseudo-compositional model (ECLIPSE 100)

- **BGGI** Variation of saturated gas FVF with pressure and Gi.
- **BOGI** Variation of saturated oil FVF with pressure and Gi.
- **GIALL** Variation of saturated properties with pressure and Gi.
- **GINODE** Gi nodal values. Required
- **RSGI** Variation of saturated GOR with pressure and Gi.
- **RVGI** Variation of saturated OGR with pressure and Gi.

### Gradient option (ECLIPSE 100 special option)

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>HMMROCK</td>
<td>Cumulative multipliers on rock compressibility.</td>
</tr>
<tr>
<td></td>
<td>HMMROCKT</td>
<td>Cumulative modifiers of rock compaction parameters.</td>
</tr>
<tr>
<td></td>
<td>HMPROPS</td>
<td>Section heading for end-point scaling modifiers.</td>
</tr>
</tbody>
</table>
### Restriction | ECLIPSE 100 only keyword | Description
--- | --- | ---
| | HMROCK | Calculate gradients with respect to rock compressibility multipliers.
| | HMROCKT | Calculate gradients with respect to rock compaction parameters.
| Required if HMROCKT used. | HMRREF | Reference pressures for HMROCKT.

### Initial composition (ECLIPSE 300)
- **COMPVD** Total composition with respect to depth tables
- **XMFVP** Liquid composition with respect to pressure tables
- **YMFVP** Vapor composition with respect to pressure tables
- **ZMFVD** Total composition with respect to depth tables
- **ZI** Enter overall composition

### K Values (ECLIPSE 300)
One of **KVGEN**, **KTABLE**, **KVTABCn** is required for the **KVALUES** option.

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>KVGEN</td>
<td>Specify that K-values are to be generated internally.</td>
</tr>
<tr>
<td>KTABLE</td>
<td>Tables of K-values.</td>
</tr>
<tr>
<td>KVTABCn, KVMF, KVCOMP</td>
<td>Composition dependent K-values.</td>
</tr>
</tbody>
</table>

### Low-Salinity option (ECLIPSE 100)

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required</td>
<td>LSALTFNC</td>
<td>Tables of salt concentration versus fraction of high-salinity and low-salinity saturation functions.</td>
</tr>
</tbody>
</table>

### Miscible flood option (ECLIPSE 100)

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Miscible Flood option only</td>
<td>MSFN</td>
<td>Allows input of relative permeability curves for miscible displacement. If not entered, a linear function is used.</td>
</tr>
<tr>
<td>Miscible Flood option only</td>
<td>PMISC</td>
<td>Enables miscibility to be modeled as a function of pressure.</td>
</tr>
</tbody>
</table>
### Restriction | ECLIPSE 100 only keyword | Description
--- | --- | ---
Miscible Flood option only. Required | SDENSITY | Input of the solvent surface density.
Miscible Flood option only. | SGCWMIS | Miscible critical gas saturation tables.
Miscible Flood option only. Required | SORWMIS | Input of the residual oil saturation for the miscible displacement.
Miscible Flood option only. Required | TLMIXPAR | Input of the Todd-Longstaff mixing parameter.

## Molecular diffusion option

### ECLIPSE 100 only keywords

| Restriction | ECLIPSE 100 only keyword | Description |
--- | --- | ---
Molecular Diffusion option only. Required. | DIFFC | Molecular diffusion coefficients for each PVT region. |
Relevant only for Molecular Diffusion option in a dual porosity run | DIFFDP | Restricts the molecular diffusion calculation to just the matrix-fracture flows in dual porosity runs. |

### ECLIPSE 300 only keywords

| Restriction | ECLIPSE 300 only keyword | Description |
--- | --- | ---
Molecular Diffusion option only. Required. | DIFFAGAS | Activity corrected gas diffusion coefficients. |
Molecular Diffusion option only. Required. | DIFFAOIL | Activity corrected water diffusion coefficients. |
Molecular Diffusion option only. Required. | DIFFCGAS | Gas diffusion coefficients. |
Molecular Diffusion option only. Required. | DIFFCOIL | Oil diffusion coefficients |
Molecular Diffusion option in connection with the CO2STORE, GASWAT and GASSOL options. Required | DIFFCWAT | Water components diffusion coefficients. |
Molecular Diffusion option in connection with the CO2STORE or GASWAT option. | DIFFCWG | Water-gas cross phase diffusion coefficients. |
Molecular Diffusion option in connection with the CO2STORE or GASWAT option. | DIFFCGW | Gas-water cross phase diffusion coefficients. |
### Molecular Diffusion

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Molecular Diffusion option only.</td>
<td>DIFFCOG</td>
<td>Oil-gas cross phase diffusion coefficients.</td>
</tr>
<tr>
<td>Molecular Diffusion option only.</td>
<td>DIFFCGO</td>
<td>Gas-oil cross phase diffusion coefficients.</td>
</tr>
<tr>
<td></td>
<td>DIFFTGAS</td>
<td>Gas thermal diffusion coefficients.</td>
</tr>
<tr>
<td></td>
<td>DIFFTOIL</td>
<td>Oil thermal diffusion coefficients.</td>
</tr>
</tbody>
</table>

### NGLs (ECLIPSE 300)

- **DNGL** Partial densities for NGLs.

### Non-Darcy Flow (ECLIPSE 300)

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Non-Darcy Flow option. Required</td>
<td>VDFLOW</td>
<td>Specify velocity dependent flow coefficient.</td>
</tr>
<tr>
<td>Non-Darcy Flow option. Required</td>
<td>VDFLOWR</td>
<td>Specify velocity dependent flow coefficient for each region.</td>
</tr>
<tr>
<td>optional if item 5 of VELDEF is non-zero</td>
<td>VDKRG</td>
<td>Gas velocity dependent relative permeability data. Relevant only for gas velocity dependent relative permeabilities.</td>
</tr>
<tr>
<td></td>
<td>VDKRGC</td>
<td>Gas velocity dependent relative permeability data.</td>
</tr>
<tr>
<td>Relevant only for oil velocity dependent relative permeabilities.</td>
<td>VDKRO</td>
<td>Oil velocity dependent relative permeability data.</td>
</tr>
</tbody>
</table>

### Output

- **RPTPROPS** Report levels for PROPS data.
- **FILLEPS** Controls the output of end point data in the INIT file.

### Polymer Flood Model

#### ECLIPSE 100 Polymer flood keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 Polymer flood keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required.</td>
<td>PLMIXPAR</td>
<td>Todd-Longstaff mixing data for the Polymer Model.</td>
</tr>
<tr>
<td></td>
<td>PLYADS</td>
<td>Polymer adsorption functions (tabulated).</td>
</tr>
<tr>
<td></td>
<td>PLYADSS</td>
<td>Polymer adsorption functions varying with salinity (tabulated).</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 100 Polymer flood keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-------------</td>
<td>----------------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td>Required</td>
<td>ADSORP</td>
<td>Polymer adsorption functions (analytical) with optional salinity dependence.</td>
</tr>
<tr>
<td></td>
<td>PLYMAX</td>
<td>Polymer/salt concentrations for mixing calculations.</td>
</tr>
<tr>
<td></td>
<td>PLYROCK</td>
<td>Polymer-rock properties.</td>
</tr>
<tr>
<td></td>
<td>PLYSHEAR</td>
<td>Polymer shear thinning/thickening data.</td>
</tr>
<tr>
<td></td>
<td>PLYVISC</td>
<td>Polymer solution viscosity function.</td>
</tr>
<tr>
<td></td>
<td>PLYVISCS</td>
<td>Polymer/salt solution viscosity function.</td>
</tr>
<tr>
<td></td>
<td>SALTNODE</td>
<td>Salt concentration nodes for polymer solution viscosity.</td>
</tr>
<tr>
<td></td>
<td>ADSALNOD</td>
<td>Salt concentration nodes for polymer solution adsorption.</td>
</tr>
<tr>
<td></td>
<td>PLYATEMP</td>
<td>Polymer adsorption functions varying with temperature (tabulated).</td>
</tr>
</tbody>
</table>

**ECLIPSE 300 Polymer flood keywords**

- **PLYOPTS** Options for the Polymer Model.
- **PLYVISC** Polymer solution viscosity function.
- **PLYVISCS** Polymer/salt solution viscosity function.
- **ESPNODE** Effective salinity nodes for polymer solution viscosity.
- **SALTEFF** Effective salinity coefficients for effective salinity calculations.

**Relative permeability and capillary pressure data**

**Common keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SGOF or SLGOF required with Family (i) in systems with both gas and oil.</td>
<td>SGOF</td>
<td>Gas relative permeability and capillary pressure and oil relative permeability in gas at connate water, as functions of $S_g$.</td>
</tr>
<tr>
<td>SGOF or SLGOF required with Family (i) in systems with both gas and oil.</td>
<td>SLGOF</td>
<td>Gas relative permeability and capillary pressure and oil relative permeability in gas at connate water, as functions of $S_{lh}$.</td>
</tr>
<tr>
<td>Required with Family (i) in systems with both water and oil.</td>
<td>SWOF</td>
<td>Water relative permeability and capillary pressure oil relative permeability in water, as functions of $S_w$.</td>
</tr>
<tr>
<td>Required with Family (ii) in two or three-phase systems with gas, unless SGWFN is used in a gas-water run.</td>
<td>SGFN</td>
<td>Gas relative permeability and capillary pressure as functions of $S_g$.</td>
</tr>
<tr>
<td>Restriction</td>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------------------------------------------------------------</td>
<td>---------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Required with Family (ii) in two or three-phase systems with water,</td>
<td>SWFN</td>
<td>Water relative permeability and capillary pressure as functions of $S_w$.</td>
</tr>
<tr>
<td>unless SGWFN is used in a gas-water run.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Required with Family (ii) for two-phase systems with oil.</td>
<td>SOF2</td>
<td>Oil relative permeability as a function of $S_o$.</td>
</tr>
<tr>
<td>SOF3 or SOF32D required with Family (ii) for three-phase systems.</td>
<td>SOF3</td>
<td>Oil relative permeability as a function of $S_o$.</td>
</tr>
<tr>
<td></td>
<td>STONE</td>
<td>Use Stone’s modified formula number 2 to calculate the three phase oil relative permeability.</td>
</tr>
<tr>
<td></td>
<td>STONE1</td>
<td>Use Stone’s modified formula number 1 to calculate the three phase oil relative permeability.</td>
</tr>
<tr>
<td></td>
<td>STONE1EX</td>
<td>Apply an exponent to the saturation terms in Stone’s modified formula number 1 method for calculating the three phase oil relative permeability.</td>
</tr>
<tr>
<td></td>
<td>STONE2</td>
<td>Use Stone’s modified formula number 2 to calculate the three phase oil relative permeability.</td>
</tr>
<tr>
<td>Optional, for STONE1 method only.</td>
<td>SOMGAS</td>
<td>Specifies the minimum oil saturation used in the STONE1 method as a function of the gas saturation.</td>
</tr>
<tr>
<td>Optional, for STONE1 method only.</td>
<td>SOMWAT</td>
<td>Specifies the minimum oil saturation used in the STONE1 method as a function of the water saturation.</td>
</tr>
<tr>
<td></td>
<td>SGF32D</td>
<td>Gas relative permeability (for three-phase flows) as 2-D table.</td>
</tr>
<tr>
<td></td>
<td>SWF32D</td>
<td>Water relative permeability (for three-phase flows) as 2-D table.</td>
</tr>
<tr>
<td>Optional, for Hysteresis option only.</td>
<td>WAGHYSTR</td>
<td>Activates the WAG Hysteresis option.</td>
</tr>
</tbody>
</table>

**ECLIPSE 100 only keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optional for runs involving only gas and water. (The gas and water relative permeability and capillary pressure data could alternatively be supplied using SWFN and SGFN, allocating the capillary pressure to the water phase.)</td>
<td>SGWFN</td>
<td>Gas-water relative permeability and capillary pressure, as functions of $S_g$.</td>
</tr>
<tr>
<td>SOF3 or SOF32D required with Family (ii) for three-phase systems</td>
<td>SOF32D</td>
<td>Oil relative permeability as functions of $S_w$ and $S_g$ in a 2D table.</td>
</tr>
<tr>
<td>Optional, if SURFTENS is specified in keyword SATOPTS, or if the Drift Flux Model is used in multisegment wells.</td>
<td>STOG</td>
<td>Tables of oil/gas surface tension.</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 100 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------------------------------------------------------------</td>
<td>--------------------------</td>
<td>--------------------------------------------------</td>
</tr>
<tr>
<td>Optional, if SURFTENS is specified in keyword SATOPTS, or if the Drift Flux Model is used in multisegment wells.</td>
<td>STOW</td>
<td>Tables of oil/water surface tension.</td>
</tr>
<tr>
<td>Optional, if the Drift Flux Model is used in multisegment wells.</td>
<td>STWG</td>
<td>Tables of water/gas surface tension.</td>
</tr>
<tr>
<td>Required for Hysteresis option.</td>
<td>EHYSTR</td>
<td>Curvature parameters and model selection for hysteresis.</td>
</tr>
<tr>
<td>Optional, for Hysteresis option only</td>
<td>EHYSTRR</td>
<td>Hysteresis parameters by region.</td>
</tr>
<tr>
<td>Optional, for Hysteresis option only.</td>
<td>HYSTCHCK</td>
<td>Activates additional consistency checks between drainage and imbibition curve endpoints when using the Hysteresis option.</td>
</tr>
</tbody>
</table>

**ECLIPSE 300 only keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRCOEF option.</td>
<td>ALPHA</td>
<td>Transport coefficient tables.</td>
</tr>
<tr>
<td>TRCOEF option</td>
<td>ALPHAD</td>
<td>Transport coefficient tables for drainage processes,</td>
</tr>
<tr>
<td>TRCOEF option.</td>
<td>ALPHAI</td>
<td>Transport coefficient tables for imbibition processes,</td>
</tr>
<tr>
<td>IKU3P</td>
<td></td>
<td>Requests IKU three-phase relative permeabilities.</td>
</tr>
<tr>
<td>ODD3P</td>
<td></td>
<td>Requests ODD3P three-phase relative permeability and capillary pressure hysteresis model.</td>
</tr>
<tr>
<td>OPTODD3P</td>
<td></td>
<td>Provides backwards compatibility and saturation switching options for the ODD3P three-phase relative permeability and capillary pressure hysteresis model</td>
</tr>
<tr>
<td>BAKER1</td>
<td></td>
<td>Requests BAKER1 three-phase relative permeabilities</td>
</tr>
<tr>
<td>BAKER2</td>
<td></td>
<td>Requests BAKER2 three-phase relative permeabilities.</td>
</tr>
<tr>
<td>STONEPAR</td>
<td></td>
<td>Modify Stone’s number 1 relative permeability model</td>
</tr>
<tr>
<td>Optional, if the IKU three-phase Relative Permeability Model is requested with keyword IKU3P. Required for the ODD3P option.</td>
<td>SGF3</td>
<td>Three phase gas saturation functions.</td>
</tr>
</tbody>
</table>
### Restrictions

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 Only Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optional, if the IKU three-phase Relative Permeability Model is requested with keyword IKU3P. Required for the ODD3P option.</td>
<td><strong>SWF3</strong></td>
<td>Three phase water saturation functions.</td>
</tr>
<tr>
<td>Optional if HYSTK is selected.</td>
<td><strong>DRAINAGE</strong></td>
<td>Forces the Killough relative permeability scanning curve to be below the drainage curve.</td>
</tr>
<tr>
<td>Optional, for CO2SOL, H2SSOL and GASSOL options only.</td>
<td><strong>MIXWPERM</strong></td>
<td>Activates the mixed water relative permeability option to take into account the dissolution of gases.</td>
</tr>
<tr>
<td>Must be used when MIXWPERM is specified.</td>
<td><strong>MIXACTO</strong></td>
<td>Specifies the oil activity functions for soluble components.</td>
</tr>
<tr>
<td>Must be used when MIXWPERM is specified</td>
<td><strong>MIXACTG</strong></td>
<td>Specifies the gas activity functions for soluble components.</td>
</tr>
<tr>
<td></td>
<td><strong>SOR</strong></td>
<td>Residual oil saturation.</td>
</tr>
<tr>
<td></td>
<td><strong>SOROPTS</strong></td>
<td>Residual oil model options.</td>
</tr>
</tbody>
</table>

### Rivers (ECLIPSE 100)

- **RIVRXSEC** Defines rivers’ cross-section
- **RBEDCONT** Defines contact areas between river bed and the grid blocks.
- **DEPTHTAB** Depth data for boundary conditions.
- **MASSFLOW** Massflow data for boundary conditions.
- **QHRATING** Rating data for boundary conditions.

### Rock compressibility and compaction data

#### Common keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Overburden pressure tables</td>
<td><strong>OVERBURD</strong></td>
<td>Overburden pressure tables</td>
</tr>
<tr>
<td>Allows directional transmissibility multipliers to be defined in ROCKTAB tables</td>
<td><strong>RKTRMDIR</strong></td>
<td></td>
</tr>
<tr>
<td>Rock compaction tables Required if using standard Rock Compaction option</td>
<td><strong>ROCKTAB</strong></td>
<td>Rock compaction tables</td>
</tr>
<tr>
<td>Hysteretic rock compaction tables</td>
<td><strong>ROCKTABH</strong></td>
<td>Hysteretic rock compaction tables</td>
</tr>
<tr>
<td>Hysteretic rock compaction tables in dual porosity runs</td>
<td><strong>ROCKTHSG</strong></td>
<td>Hysteretic rock compaction tables in dual porosity runs</td>
</tr>
<tr>
<td>Rock compaction tables in dual porosity runs</td>
<td><strong>ROCKTSIG</strong></td>
<td>Rock compaction tables in dual porosity runs</td>
</tr>
<tr>
<td>Controls options associated with the rock compaction facilities</td>
<td><strong>ROCKOPTS</strong></td>
<td></td>
</tr>
<tr>
<td>Restriction</td>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------------------</td>
<td>------------------</td>
<td>--------------------------------------------------</td>
</tr>
<tr>
<td>Optional, for the Coal Bed</td>
<td>ROCKPAMA</td>
<td>Palmer-Mansoori rock model data</td>
</tr>
<tr>
<td>Methane option only</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>TPAMEPS</td>
<td>Volumetric strain values to be used by ROCKPAMA.</td>
</tr>
</tbody>
</table>

**ECLIPSE 100 only keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ROCKTABW or ROCK2D required</td>
<td>ROCKTABW</td>
<td>Water saturation dependent rock compaction tables</td>
</tr>
<tr>
<td>for water induced rock</td>
<td></td>
<td></td>
</tr>
<tr>
<td>compaction</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ROCK2D or ROCKTABW</td>
<td>ROCK2D</td>
<td>Two-dimensional water induced rock compaction</td>
</tr>
<tr>
<td>required for water</td>
<td></td>
<td></td>
</tr>
<tr>
<td>induced rock compaction</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Optional, for 2-D water</td>
<td>ROCK2DTR</td>
<td>Transmissibility modifier tables for 2-D water</td>
</tr>
<tr>
<td>induced Rock Compaction</td>
<td></td>
<td>induced compaction</td>
</tr>
<tr>
<td>only.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Required for 2-D water</td>
<td>ROCKWNOD</td>
<td>Saturation nodes for 2-D water induced rock</td>
</tr>
<tr>
<td>induced Rock Compaction</td>
<td></td>
<td>compaction tables</td>
</tr>
<tr>
<td>only.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>TPAMEPSS</td>
<td>Volumetric strain values for solvent to be used</td>
</tr>
<tr>
<td></td>
<td></td>
<td>by ROCKPAMA</td>
</tr>
</tbody>
</table>

**ECLIPSE 300 only keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RKPVMNX</td>
<td></td>
<td>Allows pore volume multiplier limits to be set</td>
</tr>
<tr>
<td></td>
<td></td>
<td>for the BOBERG model.</td>
</tr>
<tr>
<td></td>
<td>ROCK</td>
<td>Rock compressibility</td>
</tr>
<tr>
<td></td>
<td>ROCKPAME</td>
<td>Palmer-Mansoori rock model data</td>
</tr>
<tr>
<td></td>
<td>ROCKPMAT</td>
<td>Thermal Palmer-Mansoori rock model data</td>
</tr>
<tr>
<td></td>
<td>ROCKTRMX</td>
<td>Defines transmissibility variation values in the</td>
</tr>
<tr>
<td></td>
<td></td>
<td>X-direction in order to define transmissibility</td>
</tr>
<tr>
<td></td>
<td></td>
<td>as an exponential function of porosity.</td>
</tr>
<tr>
<td></td>
<td>ROCKTRMY</td>
<td>Defines transmissibility variation values in the</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Y-direction in order to define transmissibility</td>
</tr>
<tr>
<td></td>
<td></td>
<td>as an exponential function of porosity.</td>
</tr>
<tr>
<td>Description</td>
<td></td>
<td></td>
</tr>
<tr>
<td>---------------------------------------------------------------------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Defines transmissibility variation values in the Z-direction in order to define transmissibility as an exponential function of porosity.</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

## Solid option (ECLIPSE 300)

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CVTYPE</td>
<td></td>
<td>Define solid phase components</td>
</tr>
<tr>
<td>CVTYPES</td>
<td></td>
<td>Define solid phase components at surface conditions</td>
</tr>
<tr>
<td>REACSMAX</td>
<td></td>
<td>Reaction rate solid saturation limit</td>
</tr>
<tr>
<td>REACSORD</td>
<td></td>
<td>Reaction rate solid order</td>
</tr>
<tr>
<td>SCREF</td>
<td></td>
<td>Component solid phase compressibility</td>
</tr>
<tr>
<td>SDREF</td>
<td></td>
<td>Component solid phase reference density</td>
</tr>
<tr>
<td>SOLIDADS</td>
<td></td>
<td>Solid adsorption table</td>
</tr>
<tr>
<td>SOLIDMMMC</td>
<td></td>
<td>Solid mobility multiplier (as a function of adsorbed solid concentration)</td>
</tr>
<tr>
<td>SOLIDMMS</td>
<td></td>
<td>Solid mobility multiplier (as a function of adsorbed solid saturation)</td>
</tr>
<tr>
<td>SOLWTAB</td>
<td></td>
<td>Defines an interpolation parameter as a function of adsorbed solid concentration.</td>
</tr>
<tr>
<td>SOLWW</td>
<td></td>
<td>Defines a component weighting factor for calculating the wetting-solid concentration.</td>
</tr>
<tr>
<td>SPECHS</td>
<td></td>
<td>Component solid specific heat, 1st coefficient</td>
</tr>
<tr>
<td>SPECHT</td>
<td></td>
<td>Component solid specific heat, 2nd coefficient</td>
</tr>
<tr>
<td>SPREF</td>
<td></td>
<td>Component solid phase reference pressure</td>
</tr>
<tr>
<td>STREF</td>
<td></td>
<td>Component solid phase reference temperature</td>
</tr>
<tr>
<td>STHERMX1</td>
<td></td>
<td>Component solid phase thermal expansion coefficient</td>
</tr>
</tbody>
</table>

## Solvent Model (ECLIPSE 100)

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MISC</td>
<td>Miscibility function tables.</td>
</tr>
<tr>
<td>MSPN</td>
<td>Miscible saturation functions.</td>
</tr>
<tr>
<td>PMISC</td>
<td>Pressure dependent miscibility tables.</td>
</tr>
<tr>
<td>PVDS</td>
<td>PVT properties of the solvent.</td>
</tr>
<tr>
<td>SDENSITY</td>
<td>Solvent density at surface conditions.</td>
</tr>
<tr>
<td><strong>ECLIPSE 100 only keyword</strong></td>
<td><strong>Description</strong></td>
</tr>
<tr>
<td>----------------------------</td>
<td>----------------</td>
</tr>
<tr>
<td>SGCWMIS</td>
<td>Miscible critical gas saturation tables.</td>
</tr>
<tr>
<td>SSFN</td>
<td>Gas/solvent saturation functions.</td>
</tr>
</tbody>
</table>

**Surface tension effects (ECLIPSE 300)**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required</td>
<td>MISCEXP</td>
<td>Specify miscibility component.</td>
</tr>
<tr>
<td>Required</td>
<td>MISCSTR</td>
<td>Miscibility surface tension reference.</td>
</tr>
<tr>
<td>Required</td>
<td>MISCSTRP</td>
<td>Miscibility surface tension reference pressure.</td>
</tr>
<tr>
<td>Required</td>
<td>MISCSTRR</td>
<td>Allows the reference miscible surface tension to be set by saturation region number.</td>
</tr>
<tr>
<td></td>
<td>PARACHOR</td>
<td>Parachor values for surface tension evaluation.</td>
</tr>
<tr>
<td></td>
<td>STVP</td>
<td>Surface tension with respect to pressure.</td>
</tr>
</tbody>
</table>

**Surfactant option**

**ECLIPSE 100 only keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optional, for end point scaling.</td>
<td>SOCRS</td>
<td>Scaled critical oil-in-water saturations at miscible conditions.</td>
</tr>
<tr>
<td>Required if SURFSTES is not specified.</td>
<td>SURFST</td>
<td>Oil-water surface tension in the presence of surfactant.</td>
</tr>
<tr>
<td>Required if SURFST is not specified</td>
<td>SURFSTES</td>
<td>Salinity dependent oil-water surface tension in the presence of surfactant.</td>
</tr>
<tr>
<td>Required if salinity dependent surfactant model is activated by SURFSTES</td>
<td>ESSNODE</td>
<td>Salt concentration nodes for oil-water surface tension.</td>
</tr>
<tr>
<td>Required</td>
<td>SURFVISC</td>
<td>Modified water viscosity.</td>
</tr>
<tr>
<td>Required</td>
<td>SURFCAPD</td>
<td>Capillary de-saturation data.</td>
</tr>
<tr>
<td></td>
<td>SURFADS</td>
<td>Surfactant adsorption functions (tabulated).</td>
</tr>
<tr>
<td></td>
<td>ADSORP</td>
<td>Surfactant adsorption functions (analytical) with optional salinity dependence</td>
</tr>
<tr>
<td></td>
<td>SURFADDW</td>
<td>Tables of adsorbed surfactant mass versus fraction of oil-wet and water-wet curves, for calculating immiscible relative permeabilities.</td>
</tr>
<tr>
<td>Required if SURFADS is present.</td>
<td>SURFROCK</td>
<td>Rock properties.</td>
</tr>
<tr>
<td>Required for partitioned tracers.</td>
<td>TRACERKP</td>
<td>Partitioning function K(P) for partitioned tracers.</td>
</tr>
</tbody>
</table>
### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required</td>
<td>SURFST</td>
<td>Oil-water surface tension in the presence of surfactant.</td>
</tr>
<tr>
<td>Required if SURFST is not specified</td>
<td>SURFSTES</td>
<td>Salinity dependent oil-water surface tension in the presence of surfactant.</td>
</tr>
<tr>
<td>Required if salinity dependent surfactant model is activated by SURFSTES</td>
<td>ESSNODE</td>
<td>Salt concentration nodes for oil-water surface tension.</td>
</tr>
<tr>
<td>Required if salinity dependent surfactant model is activated by SURFSTES</td>
<td>SALTEFF</td>
<td>Effective salinity coefficients.</td>
</tr>
<tr>
<td>Required</td>
<td>SURFCAPD</td>
<td>Capillary de-saturation data.</td>
</tr>
<tr>
<td></td>
<td>SURFOPTS</td>
<td>Surfactant options.</td>
</tr>
</tbody>
</table>

### Temperature option

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature option only.</td>
<td>OILVISCT</td>
<td>Inputs tables of oil viscosity versus temperature.</td>
</tr>
<tr>
<td>Required for Temperature option.</td>
<td>RTEMP</td>
<td>Specify constant reservoir temperature.</td>
</tr>
<tr>
<td>Required for Temperature option.</td>
<td>RTEMPA</td>
<td>Specify constant reservoir temperature.</td>
</tr>
<tr>
<td>Required for Temperature option.</td>
<td>SPECHEAT</td>
<td>Inputs tables of fluid specific heat data.</td>
</tr>
<tr>
<td>Required for Temperature option.</td>
<td>SPECROCK</td>
<td>Inputs tables of rock specific heat data.</td>
</tr>
<tr>
<td>Temperature option only.</td>
<td>TEMPTVD</td>
<td>Requests the use of the flux limited transport scheme.</td>
</tr>
<tr>
<td>Temperature option only.</td>
<td>VISCREF</td>
<td>Sets reference conditions for viscosity versus temperature tables.</td>
</tr>
<tr>
<td>Temperature option only.</td>
<td>WATVISCT</td>
<td>Inputs tables of water viscosity versus temperature.</td>
</tr>
</tbody>
</table>

### Thermal option (ECLIPSE 300)

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Requires KWTABTn</td>
<td>AMFVD</td>
<td>Aqueous mole fractions versus depth</td>
</tr>
<tr>
<td></td>
<td>CREF</td>
<td>Component compressibilities</td>
</tr>
<tr>
<td></td>
<td>CCTYPE</td>
<td>Component liquid compressibility type</td>
</tr>
<tr>
<td></td>
<td>CCTYPES</td>
<td>Component liquid compressibility type at surface conditions</td>
</tr>
<tr>
<td></td>
<td>CGDTYPE</td>
<td>Component gas (EoS) density model</td>
</tr>
<tr>
<td></td>
<td>CGVTYPE</td>
<td>Component gas (EoS LBC) viscosity model</td>
</tr>
<tr>
<td></td>
<td>CODTYPE</td>
<td>Component oil (EoS) density model</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 300 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-------------</td>
<td>--------------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>COVTYPE</td>
<td></td>
<td>Component oil (EoS LBC) viscosity model</td>
</tr>
<tr>
<td>CVTYPE</td>
<td></td>
<td>Component volatility (dead, live, gaseous)</td>
</tr>
<tr>
<td>CVTYPES</td>
<td></td>
<td>Component volatility (dead, live, gaseous) at surface conditions</td>
</tr>
<tr>
<td>DREF</td>
<td></td>
<td>Fluid densities at the reference pressures and temperatures.</td>
</tr>
<tr>
<td>DREFS</td>
<td></td>
<td>Reference densities for use in the surface calculations</td>
</tr>
<tr>
<td>DREFT</td>
<td>alias for DREF</td>
<td></td>
</tr>
<tr>
<td>DREFTS</td>
<td>alias for DREFS</td>
<td></td>
</tr>
<tr>
<td>ENKRVT</td>
<td></td>
<td>Endpoint relative permeabilities versus temperature</td>
</tr>
<tr>
<td>ENPCVT</td>
<td></td>
<td>Endpoint capillary pressures versus temperature</td>
</tr>
<tr>
<td>ENPTVT</td>
<td></td>
<td>Endpoint saturations versus temperature</td>
</tr>
<tr>
<td>GASVISCF</td>
<td></td>
<td>Gas viscosity function of temperature</td>
</tr>
<tr>
<td>GASVISCT</td>
<td></td>
<td>Gas viscosity table as a function of temperature</td>
</tr>
<tr>
<td>HEATVAP</td>
<td></td>
<td>Component heat of vaporization at standard temperature</td>
</tr>
<tr>
<td>HEATVAPE</td>
<td></td>
<td>Component heat of vaporization</td>
</tr>
<tr>
<td>HEATVAPS</td>
<td></td>
<td>Heat of vaporization exponent</td>
</tr>
<tr>
<td>KVCR</td>
<td></td>
<td>K-value correlation parameters</td>
</tr>
<tr>
<td>KVCRS</td>
<td></td>
<td>K-value correlation parameters at surface conditions</td>
</tr>
<tr>
<td>KVGEN</td>
<td></td>
<td>K-values generated from an EoS.</td>
</tr>
<tr>
<td>KVCRWAT</td>
<td></td>
<td>Water K-value correlation parameters</td>
</tr>
<tr>
<td>KVTEMP</td>
<td></td>
<td>Temperatures for K-value tables</td>
</tr>
<tr>
<td>KWTABTn</td>
<td></td>
<td>Tables of K-values</td>
</tr>
<tr>
<td>KWTABTP</td>
<td></td>
<td>Temperatures for Kw-value tables</td>
</tr>
<tr>
<td>KVWI</td>
<td></td>
<td>Wilson K-value correlation parameters</td>
</tr>
<tr>
<td>OILCOMPR</td>
<td></td>
<td>Oil compressibility, expansion coefficient</td>
</tr>
<tr>
<td>OILMW</td>
<td></td>
<td>Oil molecular weight</td>
</tr>
<tr>
<td>OILSPECH</td>
<td></td>
<td>Oil specific heat</td>
</tr>
<tr>
<td>OILVINDEX</td>
<td></td>
<td>Oil viscosity index</td>
</tr>
<tr>
<td>OILVISCC</td>
<td></td>
<td>Oil viscosity-temperature correlations</td>
</tr>
<tr>
<td>OILVISCF</td>
<td></td>
<td>Oil viscosity function</td>
</tr>
<tr>
<td>OILVISCT</td>
<td></td>
<td>Oil viscosity table as a function of T</td>
</tr>
<tr>
<td>OILVTIM</td>
<td></td>
<td>Interpolation method for OILVISCT data</td>
</tr>
<tr>
<td>PREF</td>
<td></td>
<td>Component reference pressures</td>
</tr>
<tr>
<td>PREFS</td>
<td></td>
<td>Component reference pressures for surface EoS</td>
</tr>
<tr>
<td>PREFT</td>
<td>alias for PREF</td>
<td></td>
</tr>
</tbody>
</table>
### Tracer tracking

#### Common keywords

The following keywords can be used in both ECLIPSE 100 and ECLIPSE 300.
• **HDISP** Mechanical dispersion values.

## ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required for passive tracer tracking.</td>
<td>TRACER</td>
<td>Names and phases of passive tracers.</td>
</tr>
<tr>
<td>tracer tracking only.</td>
<td>TRACITVD</td>
<td>Requests implicit flux limiting scheme, to reduce numerical dispersion.</td>
</tr>
<tr>
<td>tracer tracking only.</td>
<td>TRACTVD</td>
<td>Requests explicit flux limiting scheme, to reduce numerical dispersion.</td>
</tr>
<tr>
<td>Required for any diffusing tracer.</td>
<td>TRDIF</td>
<td>Tracer diffusion functions.</td>
</tr>
<tr>
<td>Required for any dispersing tracer.</td>
<td>TRDIS</td>
<td>Tracer dispersion table numbers.</td>
</tr>
<tr>
<td>Required for any dispersing tracer.</td>
<td>DISPERSE</td>
<td>Tracer dispersion tables.</td>
</tr>
<tr>
<td>Optional with dispersion only.</td>
<td>DIAGDISP</td>
<td>Enables an alternate form of dispersion.</td>
</tr>
<tr>
<td>Optional with dispersion only.</td>
<td>TRNHD</td>
<td>Speeds up dispersion calculation for a tracer when the dispersion is only being used to model constant coefficient non-homogeneous diffusion.</td>
</tr>
</tbody>
</table>

## Vertical equilibrium (ECLIPSE 100)

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vertical Equilibrium option only.</td>
<td>VEFRAC</td>
<td>Fraction of VE relative permeabilities used in place of rock curves, for the entire grid.</td>
</tr>
<tr>
<td>Vertical Equilibrium option only</td>
<td>VEFRACV</td>
<td>Fraction of VE relative permeabilities used in place of rock curves, for the current box.</td>
</tr>
<tr>
<td>Vertical Equilibrium option only</td>
<td>VEFRACP</td>
<td>Fraction of VE pseudo capillary pressures used in place of rock curves, for the entire grid.</td>
</tr>
<tr>
<td>Vertical Equilibrium option only</td>
<td>VEFRACPV</td>
<td>Fraction of VE pseudo capillary pressures used in place of rock curves, for the current box.</td>
</tr>
</tbody>
</table>

## Water component option (ECLIPSE 300)

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CREFW</td>
<td>Compressibility of water components at the reference pressure.</td>
</tr>
<tr>
<td>CREFWS</td>
<td>Compressibility of water components for surface calculations.</td>
</tr>
<tr>
<td>CWTYPET</td>
<td>Water component type</td>
</tr>
<tr>
<td>DREFW</td>
<td>Density of water components at the reference pressure</td>
</tr>
<tr>
<td>ECLIPSE 300 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-------------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>DREFWS</td>
<td>Density of water components for surface calculations</td>
</tr>
<tr>
<td>MW</td>
<td>Molecular weight of water components</td>
</tr>
<tr>
<td>MWWS</td>
<td>Molecular weight of water components for surface calculations</td>
</tr>
<tr>
<td>PREFW</td>
<td>Reference pressure for water component density and viscosity calculations</td>
</tr>
<tr>
<td>PREFWS</td>
<td>Reference pressure for water component density calculations at surface conditions.</td>
</tr>
<tr>
<td>VREFW</td>
<td>Viscosity and viscosibility for water components at the reference pressure</td>
</tr>
<tr>
<td>WNAAMES</td>
<td>Water component names</td>
</tr>
</tbody>
</table>

**ODD3P three-phase relative permeability and capillary pressure hysteresis option (ECLIPSE 300)**

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>EPSODD3P</td>
<td>End-point scaling, reference surface tensions and threshold interfacial tensions</td>
</tr>
<tr>
<td>HSGRO</td>
<td>Hysteresis end-point (residual) grid-block gas saturations for a gas-oil system</td>
</tr>
<tr>
<td>HSGRW</td>
<td>Hysteresis end-point (residual) grid-block gas saturations for a gas-water system</td>
</tr>
<tr>
<td>HSORG</td>
<td>Hysteresis end-point (residual) grid-block oil saturations for an oil-gas system</td>
</tr>
<tr>
<td>HSORW</td>
<td>Hysteresis end-point (residual) grid-block oil saturations for an oil-water system</td>
</tr>
<tr>
<td>HSWRG</td>
<td>Hysteresis end-point (residual) grid-block water saturations for a water-gas system</td>
</tr>
<tr>
<td>HSWRO</td>
<td>Hysteresis end-point (residual) grid-block water saturations for a water-oil system</td>
</tr>
<tr>
<td>ODD3P</td>
<td>Select the ODD3P three-phase relative permeability and capillary pressure model</td>
</tr>
<tr>
<td>OPTODD3P</td>
<td>Provides backwards compatibility and saturation switching options for the ODD3P three-phase relative permeability and capillary pressure hysteresis model</td>
</tr>
<tr>
<td>PCODD3P</td>
<td>Capillary pressure calculation control</td>
</tr>
<tr>
<td>PCODD3PG</td>
<td>Capillary pressure gas saturation function</td>
</tr>
<tr>
<td>PCODD3PW</td>
<td>Capillary pressure water saturation function</td>
</tr>
<tr>
<td>PSGRO</td>
<td>Primary end-point (residual) grid-block gas saturations for a gas-oil system</td>
</tr>
<tr>
<td>PSGRW</td>
<td>Primary end-point (residual) grid-block gas saturations for a gas-water system</td>
</tr>
<tr>
<td>PSORG</td>
<td>Primary end-point (residual) grid-block oil saturations for an oil-gas system</td>
</tr>
<tr>
<td>PSORW</td>
<td>Primary end-point (residual) grid-block oil saturations for an oil-water system</td>
</tr>
<tr>
<td>PSWRG</td>
<td>Primary end-point (residual) grid-block water saturations for a water-gas system</td>
</tr>
<tr>
<td>PSWRO</td>
<td>Primary end-point (residual) grid-block water saturations for a water-oil system</td>
</tr>
</tbody>
</table>

**Operational keywords**

These may be used with keywords that specify arrays over grid cells.
### Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>ADD</strong></td>
<td>Adds specified constants to specified arrays in the current input box.</td>
</tr>
<tr>
<td><strong>ADDREG</strong></td>
<td>Add by MULTNUM, FLUXNUM or OPERNUM region.</td>
</tr>
<tr>
<td><strong>BOX</strong></td>
<td>Redefines the current input box.</td>
</tr>
<tr>
<td><strong>COPY</strong></td>
<td>Copies data in the current box from one specified array to another.</td>
</tr>
<tr>
<td><strong>COPYBOX</strong></td>
<td>Copies data from one box of cells to another box of the same size.</td>
</tr>
<tr>
<td><strong>COPYREG</strong></td>
<td>Copy by FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td><strong>ENDBOX</strong></td>
<td>Redefines the current input box to encompass the whole grid.</td>
</tr>
<tr>
<td><strong>ENDFIN</strong></td>
<td>End input for local grid refinement.</td>
</tr>
<tr>
<td><strong>EQUALREG</strong></td>
<td>Set by FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td><strong>EQUALS</strong></td>
<td>Sets specified arrays to specified constants within the current box.</td>
</tr>
<tr>
<td><strong>IMPORT</strong></td>
<td>Imports binary data generated by a grid pre-processor.</td>
</tr>
<tr>
<td><strong>MAXVALUE</strong></td>
<td>Applies a maximum value to specified arrays within the current box.</td>
</tr>
<tr>
<td><strong>MINVALUE</strong></td>
<td>Applies a minimum value to specified arrays within the current box.</td>
</tr>
<tr>
<td><strong>MULTIPLY</strong></td>
<td>Multiplies specified arrays by specified constants within the current box.</td>
</tr>
<tr>
<td><strong>MULTIREG</strong></td>
<td>Multiply by FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td><strong>OPERATE</strong></td>
<td>Performs arithmetic operations on arrays.</td>
</tr>
<tr>
<td><strong>OPERATER</strong></td>
<td>Performs arithmetic operations on arrays for a specific region of the grid (defined using the OPERNUM keyword in the GRID section).</td>
</tr>
<tr>
<td><strong>REFINE</strong></td>
<td>Initiates data for a named local grid.</td>
</tr>
</tbody>
</table>

### ECLIPSE 300 keywords only

- **BOUNDARY** Set area of grid to be printed.
- **LGRCOPY** Copy global values to local cells before operations on cells within an LGR.

### Alphabetic list of keywords

#### Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACTCO2S</td>
<td>Activity coefficient model for CO₂. CO2STORE only</td>
</tr>
<tr>
<td><strong>ADD</strong></td>
<td>Adds specified constants to specified arrays in the current box.</td>
</tr>
<tr>
<td><strong>ADDREG</strong></td>
<td>Adds by MULTNUM, FLUXNUM or OPERNUM region.</td>
</tr>
<tr>
<td><strong>AQUATAB</strong></td>
<td>Carter-Tracy aquifer influence functions.</td>
</tr>
<tr>
<td><strong>BOX</strong></td>
<td>Redefines the current box.</td>
</tr>
<tr>
<td><strong>COPYREG</strong></td>
<td>Copy by MULTNUM, FLUXNUM or OPERNUM region.</td>
</tr>
<tr>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>COPY</td>
<td>Copies data in the current box from one specified array to another.</td>
</tr>
<tr>
<td>COPYBOX</td>
<td>Copies data from one box of cells to another box of the same size.</td>
</tr>
<tr>
<td>DENSITY</td>
<td>Stock tank fluid densities.</td>
</tr>
<tr>
<td>ENDBOX</td>
<td>Redefines the current box to encompass the whole grid.</td>
</tr>
<tr>
<td>ENDFIN</td>
<td>End data for a local grid refinement.</td>
</tr>
<tr>
<td>ENKRVD</td>
<td>Variation of end point relative permeability values with depth.</td>
</tr>
<tr>
<td>ENPCVD</td>
<td>Variation of maximum capillary pressure values with depth.</td>
</tr>
<tr>
<td>ENPTVD</td>
<td>Variation of saturation table end points with depth.</td>
</tr>
<tr>
<td>EPSDBGS</td>
<td>Outputs scaled relative permeability curves to the Debug file for multiple sets of grid block.</td>
</tr>
<tr>
<td>EPSDEBUG</td>
<td>Outputs scaled relative permeability curves to the Debug file.</td>
</tr>
<tr>
<td>EQLDKVCR</td>
<td>K-values for equilibrium deviation reactions.</td>
</tr>
<tr>
<td>EQLDREAC</td>
<td>Equilibrium deviation reaction rates.</td>
</tr>
<tr>
<td>EQUALREG</td>
<td>Set by MULTNUM, FLUXNUM or OPERNUM region.</td>
</tr>
<tr>
<td>EQUALS</td>
<td>Sets specified arrays to specified constants in the current box.</td>
</tr>
<tr>
<td>ESSNODE</td>
<td>Effective salinity nodes for salt sensitive surfactant flood studies.</td>
</tr>
<tr>
<td>FILEUNIT</td>
<td>Confirms the units of the data set.</td>
</tr>
<tr>
<td>FOAMFRM</td>
<td>Gas reference mobility reduction factor (functional model).</td>
</tr>
<tr>
<td>FOAMFSC</td>
<td>Gas mobility reduction dependence upon surfactant concentration (functional model).</td>
</tr>
<tr>
<td>FOAMFSO</td>
<td>Gas mobility reduction dependence upon oil saturation (functional model).</td>
</tr>
<tr>
<td>FOAMFST</td>
<td>Gas-water surface tension dependence upon concentration (functional model).</td>
</tr>
<tr>
<td>FOAMFSW</td>
<td>Gas mobility reduction dependence upon water saturation (functional model).</td>
</tr>
<tr>
<td>GRAVCONS</td>
<td>Sets a user-defined gravity constant.</td>
</tr>
<tr>
<td>GRAVITY</td>
<td>Stock tank fluid gravities.</td>
</tr>
<tr>
<td>IKRG, IKRGR, IKRW, IKRWR, IKRO, IKRORG, IKRORW</td>
<td>Imbibition curve end point relative permeability values, equivalent to KRG for example.</td>
</tr>
<tr>
<td>IMPORT</td>
<td>Imports binary data generated by a grid pre-processor.</td>
</tr>
<tr>
<td>KRG</td>
<td>Scales gas relative permeability values at maximum gas saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>KRGR</td>
<td>Scales gas relative permeability values at residual oil saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>KRO</td>
<td>Scales oil relative permeability values at maximum oil saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>KRORG</td>
<td>Scales oil relative permeability values at critical gas saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>----------</td>
<td>-------------</td>
</tr>
<tr>
<td>KRO_RW</td>
<td>Scales oil relative permeability values at critical water saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>KRW</td>
<td>Scales water relative permeability values at maximum water saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>KRW_R</td>
<td>Scales water relative permeability values at residual oil saturation, for cells in the current input box.</td>
</tr>
<tr>
<td>LCUNIT</td>
<td>Set units for linear combination of phases.</td>
</tr>
<tr>
<td>MAXVALUE</td>
<td>Applies a maximum value to specified arrays within the current box.</td>
</tr>
<tr>
<td>MINVALUE</td>
<td>Applies a minimum value to specified arrays within the current box.</td>
</tr>
<tr>
<td>MULTIPLY</td>
<td>Multiplies specified arrays by constant values in the current box.</td>
</tr>
<tr>
<td>MULTI_REG</td>
<td>Multiply by MULTNUM, FLUXNUM or OPERNUM region.</td>
</tr>
<tr>
<td>OILVISCT</td>
<td>Tables of oil viscosity versus temperature.</td>
</tr>
<tr>
<td>OILVTIM</td>
<td>Interpolation method for OILVISCT data.</td>
</tr>
<tr>
<td>OPERATE</td>
<td>Performs arithmetic operations on arrays.</td>
</tr>
<tr>
<td>OPERATER</td>
<td>Performs arithmetic operations on arrays for a specific region of the grid (defined using the OPERNUM keyword in the GRID section).</td>
</tr>
<tr>
<td>OVERBURD</td>
<td>Overburden pressure tables.</td>
</tr>
<tr>
<td>PCRITS</td>
<td>Critical pressures for surface EoS.</td>
</tr>
<tr>
<td>PCW</td>
<td>Scales the maximum water capillary pressure, for cells in the current input box.</td>
</tr>
<tr>
<td>PLYVISC</td>
<td>Polymer solution viscosity function.</td>
</tr>
<tr>
<td>PLYVISCS</td>
<td>Polymer/salt solution viscosity function.</td>
</tr>
<tr>
<td>PMAX</td>
<td>Maximum pressure expected in the simulation.</td>
</tr>
<tr>
<td>PPCWMAX</td>
<td>Limits the calculated PCW values when the SWATINIT option is used.</td>
</tr>
<tr>
<td>PVCO</td>
<td>FVF and viscosity of live oil as functions of bubble point pressure.</td>
</tr>
<tr>
<td>PVDO</td>
<td>FVF and viscosity of dead oil as functions of pressure.</td>
</tr>
<tr>
<td>PVTW</td>
<td>FVF, compressibility and viscosity of water.</td>
</tr>
<tr>
<td>PVZG</td>
<td>Z-factor and viscosity of dry gas as functions of pressure.</td>
</tr>
<tr>
<td>REFINE</td>
<td>Specify data for local grid refinement.</td>
</tr>
<tr>
<td>RKTRMDIR</td>
<td>Allows directional transmissibility multipliers to be defined in ROCKTAB tables.</td>
</tr>
<tr>
<td>ROCK2D</td>
<td>Two-dimensional water induced rock compaction tables.</td>
</tr>
<tr>
<td>ROCKOPTS</td>
<td>Controls options associated with the rock compaction facilities.</td>
</tr>
<tr>
<td>RPTPROPS</td>
<td>Report levels for PROPS data.</td>
</tr>
<tr>
<td>RTEMP</td>
<td>Specifies a constant reservoir temperature.</td>
</tr>
<tr>
<td>SCALECRS</td>
<td>Requests alternative end point scaling method.</td>
</tr>
<tr>
<td>SGCR</td>
<td>Critical gas saturation for grid cells in the current input box.</td>
</tr>
<tr>
<td>SGF32D</td>
<td>Gas relative permeability (for three-phase flows) as a 2D table.</td>
</tr>
<tr>
<td>SGFN</td>
<td>Gas relative permeability and capillary pressure as functions of $S_g$.</td>
</tr>
<tr>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>SGL</td>
<td>Connate gas saturation for grid cells in the current input box.</td>
</tr>
<tr>
<td>SGOF</td>
<td>Gas relative permeability and cap pressure, and oil relative permeability in gas, versus $S_g$.</td>
</tr>
<tr>
<td>SGU</td>
<td>Maximum gas saturation for grid cells in the current input box.</td>
</tr>
<tr>
<td>SLGOF</td>
<td>Gas relative permeability and cap pressure, and oil relative permeability in gas, versus $S_{\text{liq}}$.</td>
</tr>
<tr>
<td>SOF2</td>
<td>Oil relative permeability as a function of $S_o$ in two phase system.</td>
</tr>
<tr>
<td>SOF3</td>
<td>Oil relative permeability as a function of $S_o$ in three phase system.</td>
</tr>
<tr>
<td>SOF32D</td>
<td>Oil relative permeability as a 2D function of $S_w$ and $S_g$ in three phase systems.</td>
</tr>
<tr>
<td>SOMGAS</td>
<td>Minimum oil saturation for STONE1 method, as a function of the gas saturation.</td>
</tr>
<tr>
<td>SOMWAT</td>
<td>Minimum oil saturation for STONE1 method, as a function of the water saturation.</td>
</tr>
<tr>
<td>SOWCR</td>
<td>Residual oil-in-water saturation for cells in the current box.</td>
</tr>
<tr>
<td>STONE</td>
<td>Stone’s formula for three phase oil relative permeability.</td>
</tr>
<tr>
<td>STONE1</td>
<td>Stone’s formula 1 for three phase oil relative permeability.</td>
</tr>
<tr>
<td>STONE1EX</td>
<td>Exponent applied to saturation terms in Stone’s formula 1 for three phase oil relative permeability.</td>
</tr>
<tr>
<td>STONE2</td>
<td>Stone’s formula 2 for three phase oil relative permeability.</td>
</tr>
<tr>
<td>SURFST</td>
<td>Oil-water surface tension in the presence of surfactant.</td>
</tr>
<tr>
<td>SURFCAPD</td>
<td>Capillary de-saturation data.</td>
</tr>
<tr>
<td>SWATINIT</td>
<td>Initial water saturation, for cells in the current input box, to scale the water $P_c$ curves.</td>
</tr>
<tr>
<td>SWCR</td>
<td>Critical water saturation for grid cells in the current input box.</td>
</tr>
<tr>
<td>SWF32D</td>
<td>Water relative permeability (for three-phase flows) as 2-D table.</td>
</tr>
<tr>
<td>SWFN</td>
<td>Water relative permeability and capillary pressure as functions of $S_o$.</td>
</tr>
<tr>
<td>SWL</td>
<td>Connate water saturation for grid cells in the current input box.</td>
</tr>
<tr>
<td>SWOF</td>
<td>Water relative permeability and cap pressure, and oil relative permeability in water, versus $S_w$.</td>
</tr>
<tr>
<td>SWU</td>
<td>Maximum water saturation for grid cells in the current input box.</td>
</tr>
<tr>
<td>TOLCRIT</td>
<td>Controls the method of determining the table critical saturations in pathological end point scaling cases.</td>
</tr>
<tr>
<td>TPAMEPS</td>
<td>Volumetric strain values in ROCKPAMA.</td>
</tr>
<tr>
<td>TRACER</td>
<td>Names and phases of passive tracers.</td>
</tr>
<tr>
<td>TZONE</td>
<td>Controls the variation of critical end-point with depth.</td>
</tr>
<tr>
<td>VISCREF</td>
<td>Reference conditions for viscosity versus temperature tables.</td>
</tr>
<tr>
<td>WATVISCT</td>
<td>Tables of water viscosity versus temperature.</td>
</tr>
</tbody>
</table>
## ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADSORP</td>
<td>Analytical adsorption functions.</td>
</tr>
<tr>
<td>ALKADS</td>
<td>Alkaline adsorption functions.</td>
</tr>
<tr>
<td>ALKROCK</td>
<td>Alkaline-rock properties.</td>
</tr>
<tr>
<td>ALPHAI</td>
<td>Transport coefficient tables for imbibition processes.</td>
</tr>
<tr>
<td>ALPOLADS</td>
<td>Polymer adsorption multipliers as a function of alkaline concentration.</td>
</tr>
<tr>
<td>ALSURFAD</td>
<td>Surfactant adsorption multipliers as a function of alkaline concentration.</td>
</tr>
<tr>
<td>ALSURFST</td>
<td>Oil/water surface tension multipliers as a function of alkaline concentration.</td>
</tr>
<tr>
<td>APIGROUP</td>
<td>Allows different groups of oil PVT tables to be used in different PVNUM regions when using API Tracking.</td>
</tr>
<tr>
<td>BGGI</td>
<td>Variation of saturated gas FVF with pressure and Gi.</td>
</tr>
<tr>
<td>BOGI</td>
<td>Variation of saturated oil FVF with pressure and Gi.</td>
</tr>
<tr>
<td>DEPTHTAB</td>
<td>Depth data for boundary conditions.</td>
</tr>
<tr>
<td>DIFFC</td>
<td>Molecular diffusion coefficients for each PVT region.</td>
</tr>
<tr>
<td>DIFFDP</td>
<td>Restricts the molecular diffusion calculation to just the matrix-fracture flows in dual porosity runs.</td>
</tr>
<tr>
<td>DPKRMODE</td>
<td>Modifies the oil relative permeability in dual porosity runs.</td>
</tr>
<tr>
<td>EHYSTR</td>
<td>Curvature parameters and model selection for hysteresis.</td>
</tr>
<tr>
<td>EHYSTRR</td>
<td>Hysteresis parameters by region.</td>
</tr>
<tr>
<td>ENSPCVDV</td>
<td>Variation of the scaled connate saturation, for P_c curves only, with depth.</td>
</tr>
<tr>
<td>FILLEPS</td>
<td>Controls the output of end point data in the INIT file.</td>
</tr>
<tr>
<td>FOAMADS</td>
<td>Foam adsorption functions.</td>
</tr>
<tr>
<td>FOAMFCN</td>
<td>Gas mobility reduction dependence upon capillary number (functional model).</td>
</tr>
<tr>
<td>FOAMDCYO</td>
<td>Foam decay as a function of oil saturation (tabular model).</td>
</tr>
<tr>
<td>FOAMDCYW</td>
<td>Foam decay as a function of water saturation (tabular model).</td>
</tr>
<tr>
<td>FOAMMOB</td>
<td>Gas mobility reduction data (tabular model).</td>
</tr>
<tr>
<td>FOAMMOBP</td>
<td>Pressure dependence on foam mobility reduction (tabular model).</td>
</tr>
<tr>
<td>FOAMMOBS</td>
<td>Shear dependence on foam mobility reduction (tabular model).</td>
</tr>
<tr>
<td>FOAMOPTS</td>
<td>Specifies the Foam Model options.</td>
</tr>
<tr>
<td>FOAMROCK</td>
<td>Specifies the foam rock properties.</td>
</tr>
<tr>
<td>GIALL</td>
<td>Variation of saturated properties with pressure and Gi.</td>
</tr>
<tr>
<td>GINODE</td>
<td>Gi nodal values.</td>
</tr>
<tr>
<td>HMMROCK</td>
<td>Cumulative multipliers on rock compressibility.</td>
</tr>
<tr>
<td>HMMROCKT</td>
<td>Cumulative modifiers of rock compaction parameters.</td>
</tr>
<tr>
<td>HMPROPS</td>
<td>Section heading for end-point modifiers.</td>
</tr>
<tr>
<td><strong>ECLIPSE 100 only keyword</strong></td>
<td><strong>Description</strong></td>
</tr>
<tr>
<td>-----------------------------</td>
<td>----------------</td>
</tr>
<tr>
<td>HMROCK</td>
<td>Calculate gradients with respect to rock compressibility multipliers.</td>
</tr>
<tr>
<td>HMROCKT</td>
<td>Calculate gradients with respect to rock compaction parameters.</td>
</tr>
<tr>
<td>HMRREF</td>
<td>Reference pressures for HMROCKT.</td>
</tr>
<tr>
<td>HYDRHEAD</td>
<td>Reference quantities for hydraulic head calculation.</td>
</tr>
<tr>
<td>HYDRO</td>
<td>Defines hydrocarbon type.</td>
</tr>
<tr>
<td>HYMOBGDR</td>
<td>Change the way secondary drainage curves are computed in hysteresis runs with soluble gas.</td>
</tr>
<tr>
<td>HYSTCHCK</td>
<td>Activates additional consistency checks between drainage and imbibition curve endpoints when using the Hysteresis option.</td>
</tr>
<tr>
<td>IMKRVD</td>
<td>Imbibition end point relative permeabilities versus depth.</td>
</tr>
<tr>
<td>IMPCVD</td>
<td>Variation of maximum capillary pressure values with depth for the imbibition curve.</td>
</tr>
<tr>
<td>IMPTVD</td>
<td>Imbibition end point saturation values versus depth.</td>
</tr>
<tr>
<td>IMSPCVD</td>
<td>Variation of the scaled connate saturation, for $P_c$ curves only, with depth for the imbibition curve.</td>
</tr>
<tr>
<td>INTPC</td>
<td>Invokes the dual porosity integrated $P_c$ option.</td>
</tr>
<tr>
<td>IONXROCK</td>
<td>Rock ion exchange parameters for cation exchange.</td>
</tr>
<tr>
<td>IONXSURF</td>
<td>Surfactant micelle ion exchange parameters for cation exchange.</td>
</tr>
<tr>
<td>IPCG, IPCW</td>
<td>Imbibition curve end point capillary pressure values, equivalent to $PCG$, $PCW$. Imbibition curve end point scaling data, equivalent to $SGL$ etc.</td>
</tr>
<tr>
<td>ISGLPC, ISWLPC</td>
<td>Imbibition curve end point scaling data, equivalent to $SGLPC$ and $SWLPC$.</td>
</tr>
<tr>
<td>LSALTFNC</td>
<td>Specifies tables of salt concentration versus weighting factor for high-salinity/low-salinity saturation functions.</td>
</tr>
<tr>
<td>MASSFLOW</td>
<td>Massflow data for boundary conditions.</td>
</tr>
<tr>
<td>NOWARNEP</td>
<td>Suppresses all warning messages resulting from end point consistency checks</td>
</tr>
<tr>
<td>PECOEFS</td>
<td>Specifies petro-elastic fluid and rock properties, and other Petro-elastic Model options.</td>
</tr>
<tr>
<td>PEKTABx</td>
<td>Pressure dependent petro-elastic frame bulk modulus functions.</td>
</tr>
<tr>
<td>PEGTABx</td>
<td>Pressure dependent petro-elastic frame shear modulus functions.</td>
</tr>
<tr>
<td>PLMIXPAR</td>
<td>Todd-Longstaff mixing data for the Polymer model.</td>
</tr>
<tr>
<td>PLYESAL</td>
<td>Coefficients for polymer effective salinity.</td>
</tr>
<tr>
<td>REACACT</td>
<td>Reaction activation molar energy</td>
</tr>
<tr>
<td>ROCKWNOD</td>
<td>Saturation nodes for 2-D water induced rock compaction tables.</td>
</tr>
<tr>
<td>RSCONST</td>
<td>Sets a constant $R_s$ for dead oil over the whole field.</td>
</tr>
<tr>
<td>RSCONSTT</td>
<td>Sets constant $R_s$ for dead oil, a different value in each PVT region.</td>
</tr>
<tr>
<td>RSGI</td>
<td>Variation of saturated GOR with pressure and Gi.</td>
</tr>
<tr>
<td>RVGI</td>
<td>Variation of saturated OGR with pressure and Gi.</td>
</tr>
<tr>
<td>ECLIPSE 100 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>--------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td>RVCONST</td>
<td>Sets a constant $R_v$ for dry gas over the whole field.</td>
</tr>
<tr>
<td>RVCONSTT</td>
<td>Sets constant $R_v$ for dry gas, a different value in each PVT region.</td>
</tr>
<tr>
<td>SALTNODE</td>
<td>Salt concentration nodes for polymer solution viscosity.</td>
</tr>
<tr>
<td>SCALELIM</td>
<td>Sets saturation scaling limits for each end point scaling versus depth region.</td>
</tr>
<tr>
<td>SDENSITY</td>
<td>Input of the solvent surface density.</td>
</tr>
<tr>
<td>SGCWMIS</td>
<td>Miscible critical gas saturation tables.</td>
</tr>
<tr>
<td>SGLPC</td>
<td>Connate gas saturation, for Pc curves only, for cells in the current input box.</td>
</tr>
<tr>
<td>SGWPN</td>
<td>Gas-water relative permeability and capillary pressure, as functions of $S_g$.</td>
</tr>
<tr>
<td>SORWMIS</td>
<td>Input of the residual oil saturation for miscible displacement.</td>
</tr>
<tr>
<td>SPECHEAT</td>
<td>Tables of fluid specific heat data.</td>
</tr>
<tr>
<td>SPECROCK</td>
<td>Tables of rock specific heat data.</td>
</tr>
<tr>
<td>STOG</td>
<td>Tables of oil/gas surface tension.</td>
</tr>
<tr>
<td>STOW</td>
<td>Tables of oil/gas surface tension.</td>
</tr>
<tr>
<td>SURFADDW</td>
<td>Tables of water/gas surface tension.</td>
</tr>
<tr>
<td>SURFESAL</td>
<td>Tables of adsorbed surfactant mass vs. fraction of oil-wet and water-wet curves, for calculating immiscible relative permeabilities.</td>
</tr>
<tr>
<td>STWG</td>
<td>Coefficients for surfactant effective salinity.</td>
</tr>
<tr>
<td>SWLPC</td>
<td>Connate water saturation, for Pc curves only, for cells in the current input box.</td>
</tr>
<tr>
<td>TEMPLTVD</td>
<td>Requests flux limited transport for the Temperature option.</td>
</tr>
<tr>
<td>TLMIXPAR</td>
<td>Input of the Todd-Longstaff mixing parameter.</td>
</tr>
<tr>
<td>TPAMEPSS</td>
<td>Volumetric strain values for solvent in ROCKPAMA.</td>
</tr>
<tr>
<td>TRACERKM</td>
<td>Partitioning functions $K(P)$ for generalized, multi-partitioned tracers.</td>
</tr>
<tr>
<td>TRACERKP</td>
<td>Partitioning function $K(P)$ for partitioned tracers.</td>
</tr>
<tr>
<td>TRACTVD</td>
<td>Requests explicit flux limiting scheme to reduce tracer numerical dispersion.</td>
</tr>
<tr>
<td>TRACITVD</td>
<td>Requests implicit flux limiting scheme, to reduce numerical dispersion.</td>
</tr>
<tr>
<td>VEFRAC</td>
<td>Fraction of VE relative permeabilities used in place of rock curves, for the entire grid.</td>
</tr>
<tr>
<td>VEFRACP</td>
<td>Fraction of VE pseudo capillary pressures used in place of rock curves, for the entire grid.</td>
</tr>
<tr>
<td>VEFRACPV</td>
<td>Fraction of VE pseudo capillary pressures used in place of rock curves, for the current box.</td>
</tr>
<tr>
<td>VEFRACV</td>
<td>Fraction of VE relative permeabilities used in place of rock curves, for the current box.</td>
</tr>
<tr>
<td>WAGHYSTR</td>
<td>Activates the WAG Hysteresis option.</td>
</tr>
</tbody>
</table>
# ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACF</td>
<td></td>
<td>Acentric factors.</td>
</tr>
<tr>
<td>ACFDET</td>
<td></td>
<td>Acentric factor for detailed components</td>
</tr>
<tr>
<td>ACFS</td>
<td></td>
<td>Acentric factors for surface production system</td>
</tr>
<tr>
<td>ALPHA</td>
<td></td>
<td>Transport coefficient tables.</td>
</tr>
<tr>
<td>ALPHAD</td>
<td></td>
<td>Transport coefficient tables for drainage processes.</td>
</tr>
<tr>
<td>AMFVD</td>
<td></td>
<td>Aqueous mole fractions versus depth.</td>
</tr>
<tr>
<td>BIC</td>
<td></td>
<td>Binary interaction coefficients.</td>
</tr>
<tr>
<td>BICAQ1BICAQ2BICAQ3</td>
<td></td>
<td>Binary interaction coefficients for aqueous phase.</td>
</tr>
<tr>
<td>BICS</td>
<td></td>
<td>Binary interaction coefficients for surface EoS.</td>
</tr>
<tr>
<td>BIOTCTAB</td>
<td></td>
<td>Defines a multiplier for the Biot constant, this being a two-dimensional tabular function of maximum and minimum principal stress.</td>
</tr>
<tr>
<td>BOUNDARY</td>
<td></td>
<td>Set area of grid to be printed.</td>
</tr>
<tr>
<td>CALVAL</td>
<td></td>
<td>Component calorific values.</td>
</tr>
<tr>
<td>CALVALR</td>
<td></td>
<td>Component calorific values for multiple equation of state regions.</td>
</tr>
<tr>
<td>CATYPE</td>
<td></td>
<td>Component type for asphaltene modeling.</td>
</tr>
<tr>
<td>CCTYPE</td>
<td></td>
<td>Component liquid compressibility type.</td>
</tr>
<tr>
<td>CCTYPES</td>
<td></td>
<td>Component liquid compressibility type at surface conditions.</td>
</tr>
<tr>
<td>CNAME</td>
<td></td>
<td>Component names.</td>
</tr>
<tr>
<td>CGDTYPE</td>
<td></td>
<td>Component gas (EoS) density model.</td>
</tr>
<tr>
<td>CGVTYPE</td>
<td></td>
<td>Component gas (EoS LBC) viscosity model.</td>
</tr>
<tr>
<td>CODTYPE</td>
<td></td>
<td>Component oil (EoS) density model.</td>
</tr>
<tr>
<td>COVTYPE</td>
<td></td>
<td>Component oil (EoS LBC) viscosity model.</td>
</tr>
<tr>
<td>COMPVD</td>
<td></td>
<td>Total composition with respect to depth</td>
</tr>
<tr>
<td>CREF</td>
<td></td>
<td>Component compressibilities.</td>
</tr>
<tr>
<td>CREFW</td>
<td></td>
<td>Compressibility of water components at the reference pressure.</td>
</tr>
<tr>
<td>CREFWS</td>
<td></td>
<td>Compressibility of water components for surface calculations</td>
</tr>
<tr>
<td>CVTYPE</td>
<td></td>
<td>Component volatility (dead, live, gaseous).</td>
</tr>
<tr>
<td>CVTYPES</td>
<td></td>
<td>Component volatility (dead, live, gaseous) at surface conditions.</td>
</tr>
<tr>
<td>CWTYPE</td>
<td></td>
<td>Water component type.</td>
</tr>
<tr>
<td>DETAILMF</td>
<td></td>
<td>Initial make-up of a lumped component in terms of its detailed components versus depth.</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 300 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>------------------</td>
<td>--------------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td></td>
<td>DETAILVD</td>
<td>Initial make-up of a lumped component in terms of its detailed components versus depth.</td>
</tr>
<tr>
<td></td>
<td>DIFFAGAS</td>
<td>Activity corrected gas diffusion coefficients.</td>
</tr>
<tr>
<td></td>
<td>DIFFAOIL</td>
<td>Activity corrected oil diffusion coefficients.</td>
</tr>
<tr>
<td></td>
<td>DIFFCGAS</td>
<td>Gas diffusion coefficients.</td>
</tr>
<tr>
<td></td>
<td>DIFFCGO</td>
<td>Gas-Oil cross phase diffusion coefficients.</td>
</tr>
<tr>
<td>CO2STORE or GASWAT</td>
<td>DIFFCGW</td>
<td>Gas-Water cross phase diffusion coefficients.</td>
</tr>
<tr>
<td></td>
<td>DIFFCOIL</td>
<td>Oil diffusion coefficients.</td>
</tr>
<tr>
<td></td>
<td>DIFFCOG</td>
<td>Oil-Gas cross phase diffusion coefficients.</td>
</tr>
<tr>
<td>GASWAT, CO2STORE or GASWAT</td>
<td>DIFFCWAT</td>
<td>Water diffusion coefficients.</td>
</tr>
<tr>
<td>GASWAT, CO2STORE or GASWAT</td>
<td>DIFFCWG</td>
<td>Water-Gas cross phase diffusion coefficients.</td>
</tr>
<tr>
<td></td>
<td>DIFFTGAS</td>
<td>Gas thermal diffusion coefficients.</td>
</tr>
<tr>
<td></td>
<td>DIFFTOIL</td>
<td>Oil thermal diffusion coefficients.</td>
</tr>
<tr>
<td></td>
<td>DNGL</td>
<td>Partial densities for NGLs.</td>
</tr>
<tr>
<td></td>
<td>DRAINAGE</td>
<td>Forces the Killough relative permeability scanning curve to be below the drainage curve.</td>
</tr>
<tr>
<td></td>
<td>DREF</td>
<td>Reference densities.</td>
</tr>
<tr>
<td></td>
<td>DREFS</td>
<td>Reference densities for surface EoS.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>DREFT</td>
<td>Alias for DREF.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>DREFTS</td>
<td>Alias for DREFS.</td>
</tr>
<tr>
<td></td>
<td>DREFW</td>
<td>Density of water components at the reference pressure.</td>
</tr>
<tr>
<td></td>
<td>DREFWS</td>
<td>Density of water components for surface calculations.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>ENKRVT</td>
<td>Endpoint relative permeabilities versus temperature.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>ENPCVT</td>
<td>Endpoint capillary pressures versus temperature.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>ENPTVT</td>
<td>Endpoint saturations versus temperature.</td>
</tr>
<tr>
<td></td>
<td>EOS</td>
<td>Specify which equation of state is to be used.</td>
</tr>
<tr>
<td></td>
<td>EOSS</td>
<td>Specify surface equation of state.</td>
</tr>
<tr>
<td>ODD3P option.</td>
<td>EPSODD3P</td>
<td>End-point scaling, reference surface tensions and threshold interfacial tensions.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>EQLDKVCR</td>
<td>K-values for equilibrium deviation reactions</td>
</tr>
<tr>
<td>Thermal only</td>
<td>EQLDREAC</td>
<td>Equilibrium deviation reaction rates.</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 300 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------</td>
<td>--------------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td></td>
<td>ESPNODE</td>
<td>Effective salinity nodes for salt sensitive polymer flood studies.</td>
</tr>
<tr>
<td></td>
<td>ESSNODE</td>
<td>Effective salinity nodes for salt sensitive polymer flood studies.</td>
</tr>
<tr>
<td></td>
<td>FACTLI</td>
<td>Modify Li critical temperature correlation.</td>
</tr>
<tr>
<td></td>
<td>FVST</td>
<td>Specifies miscibility variation with surface tension.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>GASVISC°F</td>
<td>Gas viscosity function of temperature.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>GASVISCT</td>
<td>Gas viscosity table as a function of temperature.</td>
</tr>
<tr>
<td>Geomechanics only</td>
<td>GEOYLD°F</td>
<td>Defines parameters for the plastic yield function in each geomechanics region.</td>
</tr>
<tr>
<td></td>
<td>GREF</td>
<td>Reference gravities for EoS.</td>
</tr>
<tr>
<td></td>
<td>GREFS</td>
<td>Reference gravities for surface EoS.</td>
</tr>
<tr>
<td></td>
<td>GSF</td>
<td>Gas saturation functions.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>HEATVAP</td>
<td>Component heat of vaporization.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>HEATVAPE</td>
<td>Heat of vaporization exponent.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>HEATVAPS</td>
<td>Component heat of vaporization at standard temperature.</td>
</tr>
<tr>
<td>ODD3P option.</td>
<td>HSGRO</td>
<td>Hysteresis end-point (residual) grid-block gas saturations for a gas-oil system.</td>
</tr>
<tr>
<td>ODD3P option.</td>
<td>HSGRW</td>
<td>Hysteresis end-point (residual) grid-block gas saturations for a gas-water system.</td>
</tr>
<tr>
<td>ODD3P option.</td>
<td>HSORG</td>
<td>Hysteresis end-point (residual) grid-block oil saturations for an oil-gas system.</td>
</tr>
<tr>
<td>ODD3P option.</td>
<td>HSORW</td>
<td>Hysteresis end-point (residual) grid-block oil saturations for an oil-water system.</td>
</tr>
<tr>
<td>ODD3P option.</td>
<td>HSWRG</td>
<td>Hysteresis end-point (residual) grid-block water saturations for a water-gas system.</td>
</tr>
<tr>
<td>ODD3P option.</td>
<td>HSWRO</td>
<td>Hysteresis end-point (residual) grid-block water saturations for a water-oil system.</td>
</tr>
<tr>
<td></td>
<td>HYDRO</td>
<td>Defines hydrocarbon type.</td>
</tr>
<tr>
<td></td>
<td>IKU3P</td>
<td>Requests IKU three-phase relative permeabilities.</td>
</tr>
<tr>
<td></td>
<td>KRGDI</td>
<td>Gas relative permeability damage initiation.</td>
</tr>
<tr>
<td></td>
<td>KRGDM</td>
<td>Gas relative permeability damage multipliers.</td>
</tr>
<tr>
<td></td>
<td>KRODI</td>
<td>Oil relative permeability damage initiation.</td>
</tr>
<tr>
<td></td>
<td>KRODM</td>
<td>Oil relative permeability damage multipliers.</td>
</tr>
<tr>
<td></td>
<td>KRWDI</td>
<td>Water relative permeability damage initiation.</td>
</tr>
<tr>
<td></td>
<td>KRWDM</td>
<td>Water relative permeability damage multipliers.</td>
</tr>
<tr>
<td></td>
<td>KVCOMP</td>
<td>Interpolation component for composition dependent K-values.</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 300 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-------------</td>
<td>--------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td>Thermal only</td>
<td>KVCRS</td>
<td>K-value correlation parameters at surface conditions.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>KVCRWAT</td>
<td>Water K-value correlation parameters.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>KVGREN</td>
<td>Specify that K-values are to be generated internally.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>KVTEMP</td>
<td>Temperatures for K-value tables.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>KVWI</td>
<td>Wilson K-value correlation parameters.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>KWTABTn</td>
<td>Tables of Kw-values.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>KWTEMP</td>
<td>Temperatures for Kw-value tables.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>LBCCOEF</td>
<td>Set non-default LBC coefficients.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>LBCCOEFR</td>
<td>Set non-default LBC coefficients for multiple equation of state regions.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>LGRCOPY</td>
<td>Copy global values to local cells before operations on cells within an LGR.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>LILIM</td>
<td>Specifies the range for Li-interpolation.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>LUMPING</td>
<td>Set up names of tracers describing detailed composition of a lumped hydrocarbon component.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>MISCEXP</td>
<td>Miscibility exponent.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>MISCSTR</td>
<td>Miscibility surface tension reference.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>MISCSTRP</td>
<td>Miscibility surface tension reference pressure.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>MISCSTRR</td>
<td>Allows the reference miscible surface tension to be set by saturation region number.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>MSFN</td>
<td>Relative permeability curves for miscible displacement.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>MW</td>
<td>Molecular weights.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>MWDETAIL</td>
<td>Molecular weights for detailed components.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>MWS</td>
<td>Molecular weights for surface EoS.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>MWWS</td>
<td>Molecular weight of water components.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>MWWS</td>
<td>Molecular weight of water components for surface calculations.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>NCOMPS</td>
<td>Confirm number of components.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>ODD3P</td>
<td>ODD3P three-phase relative permeability and capillary pressure hysteresis option.</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 300 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-------------</td>
<td>-------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td>ODD3P option</td>
<td>OPTODD3P</td>
<td>Provides backwards compatibility and saturation switching options</td>
</tr>
<tr>
<td>Thermal only</td>
<td>OILCOMPR</td>
<td>Oil compressibility, expansion coefficient.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>OILMW</td>
<td>Oil molecular weight.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>OILSPEECH</td>
<td>Oil specific heat.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>OILVINDX</td>
<td>Oil viscosity index.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>OILVISCC</td>
<td>Oil viscosity-temperature correlation</td>
</tr>
<tr>
<td>Thermal only</td>
<td>OILVISCF</td>
<td>Oil viscosity function.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>OMEGAA</td>
<td>Overrides default $\Omega_a$ values.</td>
</tr>
<tr>
<td></td>
<td>OMEGAADDE</td>
<td>Overrides default $\Omega_a$ values for detailed composition.</td>
</tr>
<tr>
<td></td>
<td>OMEGAAS</td>
<td>Overrides default surface $\Omega_a$ values.</td>
</tr>
<tr>
<td></td>
<td>OMEGAASD</td>
<td>Overrides default surface $\Omega_a$ values for detailed composition.</td>
</tr>
<tr>
<td></td>
<td>OMEGAB</td>
<td>Overrides default $\Omega_b$ values.</td>
</tr>
<tr>
<td></td>
<td>OMEGABDE</td>
<td>Overrides default $\Omega_b$ values for detailed composition.</td>
</tr>
<tr>
<td></td>
<td>OMEGABS</td>
<td>Overrides default surface $\Omega_b$ values.</td>
</tr>
<tr>
<td></td>
<td>OMEGABSD</td>
<td>Overrides default surface $\Omega_b$ values for detailed composition.</td>
</tr>
<tr>
<td></td>
<td>PARACHOR</td>
<td>Component parachors.</td>
</tr>
<tr>
<td></td>
<td>PCG</td>
<td>Scales the maximum gas capillary pressure, for cells in the current input box.</td>
</tr>
<tr>
<td></td>
<td>PCRIT</td>
<td>Critical pressures.</td>
</tr>
<tr>
<td></td>
<td>PCRITDET</td>
<td>Critical pressures for detailed components.</td>
</tr>
<tr>
<td></td>
<td>PCRITSDE</td>
<td>Critical pressures for surface EoS for detailed components.</td>
</tr>
<tr>
<td>ODD3P option</td>
<td>PCODD3P</td>
<td>Capillary pressure calculation control.</td>
</tr>
<tr>
<td>ODD3P option</td>
<td>PCODD3PG</td>
<td>Capillary pressure gas saturation function</td>
</tr>
<tr>
<td>ODD3P option</td>
<td>PCODD3PW</td>
<td>Capillary pressure water saturation function</td>
</tr>
<tr>
<td></td>
<td>PERMSTAB</td>
<td>Defines a permeability multiplier, this being a two-dimensional tabular function of maximum and minimum principal stress.</td>
</tr>
<tr>
<td></td>
<td>PLYOPTS</td>
<td>Options for the polymer model.</td>
</tr>
<tr>
<td></td>
<td>PREFS</td>
<td>Component reference pressures for surface EoS.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>PREFW</td>
<td>Reference pressure for water component density and viscosity calculations.</td>
</tr>
<tr>
<td></td>
<td>PREFWS</td>
<td>Reference pressure for water component density calculations at surface conditions.</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 300 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>--------------------</td>
<td>--------------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>ODD3P option.</td>
<td>PSGRO</td>
<td>Primary end-point (residual) grid-block gas saturations for the gas-oil system</td>
</tr>
<tr>
<td>ODD3P option.</td>
<td>PSGRW</td>
<td>Primary end-point (residual) grid-block gas saturations for the gas-water system</td>
</tr>
<tr>
<td>ODD3P option</td>
<td>PSORG</td>
<td>Primary end-point (residual) grid-block oil saturations for the oil-gas system</td>
</tr>
<tr>
<td>ODD3P option</td>
<td>PSORW</td>
<td>Primary end-point (residual) grid-block oil saturations for the oil-water system for the option.</td>
</tr>
<tr>
<td>ODD3P option</td>
<td>PSWRG</td>
<td>Primary end-point (residual) grid-block water saturations for the water-gas system</td>
</tr>
<tr>
<td>ODD3P option.</td>
<td>PSWRO</td>
<td>Primary end-point (residual) grid-block water saturations for the water-oil system</td>
</tr>
<tr>
<td></td>
<td>REACLIMS</td>
<td>Pressure and temperature limits for reaction rate calculations.</td>
</tr>
<tr>
<td></td>
<td>REACPORD</td>
<td>Reaction rate porosity order.</td>
</tr>
<tr>
<td></td>
<td>REACSMAX</td>
<td>Maximum solid saturation in chemical reaction rates.</td>
</tr>
<tr>
<td></td>
<td>REACSORORD</td>
<td>Order of solid term in chemical reaction rates.</td>
</tr>
<tr>
<td></td>
<td>RKPVMNMNX</td>
<td>Allows pore volume multiplier limits to be set for the BOBERG model.</td>
</tr>
<tr>
<td></td>
<td>ROCKTRMX</td>
<td>Define transmissibility variation values in the X-direction in order to define transmissibility as an exponential function of porosity.</td>
</tr>
<tr>
<td></td>
<td>ROCKTRMY</td>
<td>Define transmissibility variation values in the Y-direction in order to define transmissibility as an exponential function of porosity.</td>
</tr>
<tr>
<td></td>
<td>ROCKTRMZ</td>
<td>Define transmissibility variation values in the Z-direction in order to define transmissibility as an exponential function of porosity.</td>
</tr>
<tr>
<td></td>
<td>RSWVD</td>
<td>Aqueous concentration of CO\textsubscript{2} versus depth.</td>
</tr>
<tr>
<td></td>
<td>SALTEFF</td>
<td>Coefficients to calculate effective salinity</td>
</tr>
<tr>
<td>Polymer and</td>
<td>SALINITY</td>
<td>Reservoir salinity.</td>
</tr>
<tr>
<td>Surfactant models.</td>
<td>SALINITR</td>
<td>Regional reservoir salinity.</td>
</tr>
<tr>
<td>Geomechanics only</td>
<td>SCDATABG</td>
<td>Defines a scale damage table to be used together with the proppant flowback feature.</td>
</tr>
<tr>
<td>Thermal only</td>
<td>SCREF</td>
<td>Component solid phase compressibility.</td>
</tr>
<tr>
<td>Thermal</td>
<td>SDREF</td>
<td>Component solid phase reference density.</td>
</tr>
<tr>
<td>Thermal</td>
<td>SGF3</td>
<td>Three-phase gas saturation functions.</td>
</tr>
<tr>
<td></td>
<td>SOGCR</td>
<td>Residual oil-in-gas saturation for cells in the current box.</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 300 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-------------</td>
<td>--------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td>SOLIDADS</td>
<td></td>
<td>Solid adsorption table.</td>
</tr>
<tr>
<td>SOLIDMMMC</td>
<td></td>
<td>Solid mobility multiplier (as a function of adsorbed concentration).</td>
</tr>
<tr>
<td>SOLIDMMS</td>
<td></td>
<td>Solid mobility multiplier (as a function of adsorbed saturation).</td>
</tr>
<tr>
<td>SOLUBILI</td>
<td></td>
<td>Specify properties of Water-CO$_2$ systems.</td>
</tr>
<tr>
<td>SOLUBILS</td>
<td></td>
<td>Specify properties of Water-CO$_2$ systems with salt dependencies.</td>
</tr>
<tr>
<td>SOLUCO2H</td>
<td></td>
<td>Specify properties of Water-CO$_2$ systems with H$_2$S dependencies.</td>
</tr>
<tr>
<td>SOLUH2SC</td>
<td></td>
<td>Specify properties of Water-H$_2$S systems with CO$_2$ dependencies.</td>
</tr>
<tr>
<td>SOLWTAB</td>
<td></td>
<td>Defines an interpolation parameter as a function of adsorbed solid concentration.</td>
</tr>
<tr>
<td>SOLWW</td>
<td></td>
<td>Defines a component weighting factor for calculating the wetting-solid concentration.</td>
</tr>
<tr>
<td>SOR</td>
<td></td>
<td>Residual oil saturation.</td>
</tr>
<tr>
<td>SOROPTS</td>
<td></td>
<td>Residual oil model options.</td>
</tr>
<tr>
<td>SPECHA</td>
<td></td>
<td>Component oil specific heat, 1$^{st}$ coefficient.</td>
</tr>
<tr>
<td>SPECHB</td>
<td></td>
<td>Component oil specific heat, 2$^{nd}$ coefficient.</td>
</tr>
<tr>
<td>SPECHG</td>
<td></td>
<td>Component gas specific heat, 1$^{st}$ coefficient.</td>
</tr>
<tr>
<td>SPECHH</td>
<td></td>
<td>Component gas specific heat, 2$^{nd}$ coefficient.</td>
</tr>
<tr>
<td>SPECHS</td>
<td></td>
<td>Component solid specific heat, 1$^{st}$ coefficient.</td>
</tr>
<tr>
<td>SPECHT</td>
<td></td>
<td>Component solid specific heat, 2$^{nd}$ coefficient.</td>
</tr>
<tr>
<td>SPECHW1</td>
<td></td>
<td>Component aqua specific heat, 1$^{st}$ coefficient.</td>
</tr>
<tr>
<td>SPECHW2</td>
<td></td>
<td>Component aqua specific heat, 2$^{nd}$ coefficient.</td>
</tr>
<tr>
<td>SPREF</td>
<td></td>
<td>Component solid phase reference pressure.</td>
</tr>
<tr>
<td>SSHIFT</td>
<td></td>
<td>Equation of state shift parameters.</td>
</tr>
<tr>
<td>SSHIFTS</td>
<td></td>
<td>Equation of state shift parameters for surface EoS.</td>
</tr>
<tr>
<td>STCOND</td>
<td></td>
<td>Specify standard conditions.</td>
</tr>
<tr>
<td>STONEPAR</td>
<td></td>
<td>Modify Stone’s formula 1 three phase oil relative permeability</td>
</tr>
<tr>
<td>STOPROD</td>
<td></td>
<td>Stoichiometric coefficients of products.</td>
</tr>
<tr>
<td>STOREAC</td>
<td></td>
<td>Stoichiometric coefficients of reactants.</td>
</tr>
<tr>
<td>STREF</td>
<td></td>
<td>Component solid phase reference temperature.</td>
</tr>
<tr>
<td>STHERMX1</td>
<td></td>
<td>Component solid phase thermal expansion coefficient.</td>
</tr>
<tr>
<td>STVP</td>
<td></td>
<td>Surface tension with respect to pressure.</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 300 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-------------</td>
<td>--------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td></td>
<td>SURFOPTS</td>
<td>Surfactant options</td>
</tr>
<tr>
<td></td>
<td>SWF3</td>
<td>Three phase water saturation functions.</td>
</tr>
<tr>
<td></td>
<td>TBOIL</td>
<td>Component boiling points.</td>
</tr>
<tr>
<td></td>
<td>TBOILS</td>
<td>Component boiling points for surface EoS.</td>
</tr>
<tr>
<td></td>
<td>TCRIT</td>
<td>Critical temperatures.</td>
</tr>
<tr>
<td></td>
<td>TCRITDET</td>
<td>Critical temperatures for detailed components.</td>
</tr>
<tr>
<td></td>
<td>TCRITS</td>
<td>Critical temperatures for surface EoS.</td>
</tr>
<tr>
<td></td>
<td>TCRITSDE</td>
<td>Critical temperatures for surface EoS for detailed components</td>
</tr>
<tr>
<td></td>
<td>TEMPVD</td>
<td>Temperature vs. depth data.</td>
</tr>
<tr>
<td></td>
<td>THANALB</td>
<td>Analytic water and steam densities.</td>
</tr>
<tr>
<td></td>
<td>THANALH</td>
<td>Analytic water and steam enthalpy values.</td>
</tr>
<tr>
<td></td>
<td>THANALV</td>
<td>Analytic oil viscosities.</td>
</tr>
<tr>
<td></td>
<td>THERMEX1</td>
<td>Component thermal expansion coefficient.</td>
</tr>
<tr>
<td></td>
<td>THSVC</td>
<td>Steam viscosity coefficients.</td>
</tr>
<tr>
<td></td>
<td>THTABB</td>
<td>Tabular water and steam densities.</td>
</tr>
<tr>
<td></td>
<td>THWVC</td>
<td>Water viscosity coefficients.</td>
</tr>
<tr>
<td></td>
<td>TRANGE</td>
<td>Expected temperature limits.</td>
</tr>
<tr>
<td></td>
<td>TREF</td>
<td>Reference temperatures.</td>
</tr>
<tr>
<td></td>
<td>TREFS</td>
<td>Reference temperatures for surface EoS.</td>
</tr>
<tr>
<td>Thermal</td>
<td>TREFT</td>
<td>Alias for TREF</td>
</tr>
<tr>
<td>Thermal</td>
<td>TREFTS</td>
<td>Alias for TREFS.</td>
</tr>
<tr>
<td></td>
<td>VCOMPACT</td>
<td>Compact volumes.</td>
</tr>
<tr>
<td></td>
<td>VCRIT</td>
<td>Critical volumes.</td>
</tr>
<tr>
<td></td>
<td>VCRITDET</td>
<td>Critical volumes for detailed components.</td>
</tr>
<tr>
<td></td>
<td>VCRITS</td>
<td>Critical volumes for surface EoS.</td>
</tr>
<tr>
<td></td>
<td>VCRITSDE</td>
<td>Critical volumes for surface EoS for detailed components</td>
</tr>
<tr>
<td></td>
<td>VCRITVIS</td>
<td>Critical volumes for viscosity calculations.</td>
</tr>
<tr>
<td></td>
<td>VDFLOW</td>
<td>Specify velocity dependent flow coefficient.</td>
</tr>
<tr>
<td></td>
<td>VDKRO</td>
<td>Oil velocity dependent relative permeability data.</td>
</tr>
<tr>
<td></td>
<td>VDKRG</td>
<td>Gas velocity dependent relative permeability data.</td>
</tr>
<tr>
<td></td>
<td>VDKRGC</td>
<td>Gas velocity dependent relative permeability data.</td>
</tr>
<tr>
<td></td>
<td>VREFW</td>
<td>Viscosity and viscosibility for water components at the reference pressure.</td>
</tr>
<tr>
<td>Thermal</td>
<td>WATDENT</td>
<td>Water density temperature dependence.</td>
</tr>
<tr>
<td></td>
<td>WATERTAB</td>
<td>Water pressure tables.</td>
</tr>
</tbody>
</table>
### REGIONS section

The `REGIONS` section divides the computational grid into regions for:

- Calculation of saturation functions (relative permeability and capillary pressure)
- Calculation of PVT properties (fluid densities, FVFs, viscosities)
- Equilibration (setting initial pressures and saturations)
- Reporting of fluids in place and inter-region flows
- Calculation of directional relative permeabilities
- Calculation of saturation functions for imbibition (Hysteresis option)
- Calculation of `ROCKTAB` properties for the Rock Compaction option
- Calculation of initial tracer concentrations (Tracer Tracking option)
- Calculation of the saturation table end points from depth tables (for the saturation table End Point Scaling option)
- Calculation of mixture properties (Miscible Flood option)
- Specifying pressure maintenance regions.

If there is no `REGIONS` section, ECLIPSE puts all grid blocks into a single region for all the above operations.

### REGIONS section keywords

A list of all keywords that may occur in the `REGIONS` section, grouped by function, is given below, together with a brief description of their function. This is followed by an alphabetic list. A more detailed description may be found by looking up the keyword in the manual.
Keywords must start in column 1. All characters up to column 8 are significant.

Output

- **RPTREGS** Report levels for **REGIONS** data.

Commonly used keywords

- **FIPNUM** Fluid-in-Place regions.
- **SATNUM** Saturation table regions.
- **EQLNUM** Equilibration regions.
- **PVNUM** PVT data regions.

Additional sets of fluid-in-place regions

Extra sets of region numbers, in addition to standard **FIPNUM** set.

*ECLIPSE 100 only*

Regions number representing the gas, oil and water zones at equilibration.

Directional relative permeability

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Keywords should not be used unless DIRECT is selected in keyword SATOPTS.</td>
<td>KRNUMX, KRNUMY, KRNUMZ</td>
<td>Saturation table numbers for the +X, +Y, +Z faces of a grid block.</td>
</tr>
<tr>
<td>Keywords should not be used unless IRREVERS is also selected in keyword SATOPTS.</td>
<td>KRNUMX−, KRNUMY−, KRNUMZ−</td>
<td>Saturation table numbers for the -X, -Y, -Z faces of a grid block.</td>
</tr>
</tbody>
</table>

Dual porosity option (ECLIPSE 100 only)

- **KRNUMMF** Matrix-fracture flow saturation tables numbers.
- **IMBNUMMF** Matrix-fracture imbibition region numbers.

Only concerns dual porosity runs (keyword **DUALPORO** in **RUNSPEC**) using the Hysteresis option (item **HYSTER** in keyword **SATOPTS**).

Geomechanics option (ECLIPSE 300)

- **YLDNUM** Defines a geomechanics region number for each grid block.

Hysteresis option (excluding ODD3P)

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>IMBNUM</td>
<td>Imbibition saturation table numbers for all faces of every grid block in current box.</td>
</tr>
</tbody>
</table>
### WAG hysteresis option

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Keywords should not be used unless HYSTER and DIRECT are selected in keyword SATOPTS.</td>
<td>IMBNUMX, IMBNUMY, IMBNUMZ</td>
<td>Imbibition saturation table numbers for the +X, +Y, +Z face of a grid block.</td>
</tr>
<tr>
<td>Keywords should not be used unless IRREVERS is also selected in keyword SATOPTS.</td>
<td>IMBNUMX-, IMBNUMY-, IMBNUMZ-</td>
<td>Imbibition saturation table numbers for the -X, -Y, -Z faces of a grid block.</td>
</tr>
</tbody>
</table>

### Other options

#### Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ENNUM</td>
<td>End point scaling regions.</td>
</tr>
<tr>
<td>TNUM</td>
<td>Tracer tracking regions.</td>
</tr>
<tr>
<td>MISCMUM</td>
<td>Miscible Flood option regions.</td>
</tr>
<tr>
<td>OPERNUM</td>
<td>Defines region on which to do operations using the OPERATER keyword. The data can also be edited using ADDREG, EQUALREG, MULTIREG and COPYREG.</td>
</tr>
<tr>
<td>ROCKNUM</td>
<td>Rock compaction regions.</td>
</tr>
<tr>
<td>SURFNUM</td>
<td>Surfactant miscible regions.</td>
</tr>
</tbody>
</table>

#### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRKPF</td>
<td>Partitioned tracer K(P) partitioning function regions.</td>
</tr>
<tr>
<td>LSLTWNUM</td>
<td>Low-salinity (water-wet) saturation function regions.</td>
</tr>
<tr>
<td>LWSLTNUM</td>
<td>Low-salinity (oil-wet) saturation function regions.</td>
</tr>
<tr>
<td>PENUM</td>
<td>Petro-elastic Model regions.</td>
</tr>
<tr>
<td>RESIDNUM</td>
<td>Vertical Equilibrium residual flow option regions.</td>
</tr>
<tr>
<td>SURFWNUM</td>
<td>Surfactant adsorption function regions.</td>
</tr>
</tbody>
</table>
ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALPHANUM</td>
<td>Transport coefficient regions.</td>
</tr>
<tr>
<td>ALPHANUD</td>
<td>Transport coefficient regions for drainage processes.</td>
</tr>
<tr>
<td>ALPHANUI</td>
<td>Transport coefficient regions for imbibition processes.</td>
</tr>
<tr>
<td>EOSNUM</td>
<td>Equation of state regions.</td>
</tr>
<tr>
<td>PMANUM</td>
<td>Pressure maintenance regions.</td>
</tr>
<tr>
<td>SOLWNUM</td>
<td>Solid wettability regions.</td>
</tr>
<tr>
<td>THERMNUM</td>
<td>Thermal regions.</td>
</tr>
</tbody>
</table>

ODD3P three-phase relative permeability and capillary pressure hysteresis option (ECLIPSE 300)

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PSTNUM</td>
<td>Grid-block primary saturation table numbers.</td>
</tr>
<tr>
<td>ISTNUM</td>
<td>Grid-block secondary hysteresis saturation table numbers.</td>
</tr>
<tr>
<td>DSTNUM</td>
<td>Grid-block tertiary hysteresis saturation table numbers.</td>
</tr>
<tr>
<td>SDRGO</td>
<td>Gas phase initial grid-block saturation direction indicators for the gas-oil system.</td>
</tr>
<tr>
<td>SDRGW</td>
<td>Gas phase initial grid-block saturation direction indicators for the gas-water system.</td>
</tr>
<tr>
<td>SDROG</td>
<td>Oil phase initial grid-block saturation direction indicators for the gas-oil system.</td>
</tr>
<tr>
<td>SDROW</td>
<td>Oil phase initial grid-block saturation direction indicators for the oil-water system.</td>
</tr>
<tr>
<td>SDRWG</td>
<td>Water phase initial grid-block saturation direction indicators for the gas-water system.</td>
</tr>
<tr>
<td>SDRW0</td>
<td>Water phase initial grid-block saturation direction indicators for the oil-water system.</td>
</tr>
</tbody>
</table>

Operational

These may be used with keywords that specify arrays over grid cells.

Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADDREG</td>
<td>Add by FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>BOX</td>
<td>Redefines the current input box.</td>
</tr>
<tr>
<td>COPY</td>
<td>Copies data in the current box from one specified array to another.</td>
</tr>
<tr>
<td>COPYBOX</td>
<td>Copies data from one box of cells to another box of the same size.</td>
</tr>
<tr>
<td>COPYREG</td>
<td>Copy by FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>ENDBOX</td>
<td>Redefines the current input box to encompass the whole grid.</td>
</tr>
<tr>
<td>ENDFIN</td>
<td>End input for local grid refinement.</td>
</tr>
</tbody>
</table>
### ECLIPSE Reference Manual

**Data File Overview**

#### Alphabetical list of keywords

##### Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADD</td>
<td>Adds specified constants to specified arrays in the current input box.</td>
</tr>
<tr>
<td>ADDREG</td>
<td>Adds specified constants to specified arrays in a FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>BOUNDARY</td>
<td>Defines section of grid for which output maps are required.</td>
</tr>
<tr>
<td>BOX</td>
<td>Redefines the current input box.</td>
</tr>
<tr>
<td>COPY</td>
<td>Copies data in the current input box from one array to another.</td>
</tr>
<tr>
<td>COPYBOX</td>
<td>Copies data from one box of cells to another box of the same size.</td>
</tr>
<tr>
<td>COPYREG</td>
<td>Copies one array to another within a FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>ENDBOX</td>
<td>Redefines the current input box so that it covers the whole grid.</td>
</tr>
<tr>
<td>ENDNUM</td>
<td>Saturation table end point region for every grid block in the current input box.</td>
</tr>
<tr>
<td>EQLNUM</td>
<td>Equilibration region for every grid block in the current input box.</td>
</tr>
<tr>
<td>EQUALS</td>
<td>Sets specified arrays to constants within the current input box.</td>
</tr>
<tr>
<td>EQUALREG</td>
<td>Sets specified arrays to constants within a FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>FILEUNIT</td>
<td>Confirms the units of the data set.</td>
</tr>
<tr>
<td>FIPNUM</td>
<td>Standard fluid-in-place region numbers, for every grid block in the current input box.</td>
</tr>
<tr>
<td>HMxxxxxx</td>
<td>Family of history-matching regions for use with the Gradient option.</td>
</tr>
<tr>
<td>IMBNUM</td>
<td>Imbibition table region for every grid block in the current input box (if using the hysteresis option).</td>
</tr>
</tbody>
</table>

---

#### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BOUNDARY</td>
<td>Set area of grid to be printed</td>
</tr>
<tr>
<td>LGRCOPY</td>
<td>Copy global values to local cells before operations on cells within an LGR.</td>
</tr>
</tbody>
</table>

---

ECLIPSE Reference Manual
<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>IMBNUMX</td>
<td>Imbibition region for +X face of every grid block in the current input box (if using hysteresis with directional relative permeabilities).</td>
</tr>
<tr>
<td>IMBNUMX-</td>
<td>Imbibition region for -X face of every grid block in the current input box (if using hysteresis with irreversible directional relative permeabilities).</td>
</tr>
<tr>
<td>IMBNUMY</td>
<td>Imbibition region for +Y face of every grid block in the current input box (if using hysteresis with directional relative permeabilities).</td>
</tr>
<tr>
<td>IMBNUMY-</td>
<td>Imbibition region for -Y face of every grid block in the current input box (if using hysteresis with irreversible directional relative permeabilities).</td>
</tr>
<tr>
<td>IMBNUMZ</td>
<td>Imbibition region for +Z face of every grid block in the current input box (if using hysteresis with directional relative permeabilities).</td>
</tr>
<tr>
<td>IMBNUMZ-</td>
<td>Imbibition region for -Z face of every grid block in the current input box (if using hysteresis with irreversible directional relative permeabilities).</td>
</tr>
<tr>
<td>IMPORT</td>
<td>Imports binary data generated by a grid pre-processor.</td>
</tr>
<tr>
<td>KRNUMX</td>
<td>$K_r$ region for +X face of every grid block in the current input box (if using directional relative permeabilities).</td>
</tr>
<tr>
<td>KRNUMX-</td>
<td>$K_r$ region for -X face of every grid block in the current input box (if using irreversible directional relative permeabilities).</td>
</tr>
<tr>
<td>KRNUMY</td>
<td>$K_r$ region for +Y face of every grid block in the current input box (if using directional relative permeabilities).</td>
</tr>
<tr>
<td>KRNUMY-</td>
<td>$K_r$ region for -Y face of every grid block in the current input box (if using irreversible directional relative permeabilities).</td>
</tr>
<tr>
<td>KRNUMZ</td>
<td>$K_r$ region for +Z face of every grid block in the current input box (if using directional relative permeabilities).</td>
</tr>
<tr>
<td>KRNUMZ-</td>
<td>$K_r$ region for -Z face of every grid block in the current input box (if using irreversible directional relative permeabilities).</td>
</tr>
<tr>
<td>MISCNUM</td>
<td>Miscibility table region for every grid block in the current input box (if using the Miscible Flood option).</td>
</tr>
<tr>
<td>MULTIPLY</td>
<td>Multiplies specified arrays by specified constants within the current box.</td>
</tr>
<tr>
<td>MULTIREG</td>
<td>Multiply by FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>OPERATE</td>
<td>Performs arithmetic operations on arrays.</td>
</tr>
<tr>
<td>OPERATER</td>
<td>Performs arithmetic operations on arrays for a specific region of the grid.</td>
</tr>
<tr>
<td>OPERNUM</td>
<td>Defines region on which to do operations using the OPERATER keyword. The data can also be edited using ADDREG, EQUALREG, MULTIREG and COPYREG.</td>
</tr>
<tr>
<td>PVTNUM</td>
<td>PVT region for every grid block in the current input box.</td>
</tr>
<tr>
<td>ROCKNUM</td>
<td>Rock compaction table region for every grid block in the current input box (if using the Rock Compaction option).</td>
</tr>
<tr>
<td>RPTREGS</td>
<td>Switches on output of data in REGIONS section.</td>
</tr>
<tr>
<td>SATNUM</td>
<td>Saturation table region for every grid block in the current input box. (If directional relative permeability tables are being used, SATNUM regions are still used in the equilibration calculation and by the wells.)</td>
</tr>
</tbody>
</table>
### Keyword Description

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SURFNUM</td>
<td>Surfactant miscible regions</td>
</tr>
<tr>
<td>TNUM</td>
<td>Initial tracer concentration region for every grid block in the current input box (If Tracer Tracking option is activated).</td>
</tr>
<tr>
<td>WH2NUM</td>
<td>Two-phase water relative permeability saturation region numbers. This keyword is optional if WAGHYSTR has been specified but should not be specified otherwise.</td>
</tr>
<tr>
<td>WH3NUM</td>
<td>Three-phase water relative permeability saturation region numbers. This keyword is optional if WAGHYSTR has been specified but should not be specified otherwise.</td>
</tr>
</tbody>
</table>

### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FIP</td>
<td>Additional sets of fluid-in-place region numbers, for every grid block in the current input box.</td>
</tr>
<tr>
<td>FIPOWG</td>
<td>Creates a family of FIP regions to represent the gas, oil and water zones at equilibration</td>
</tr>
<tr>
<td>IMBNUMMF</td>
<td>Imbibition table region of matrix-fracture flows, for every grid block in the current input box (if using both the dual porosity and the hysteresis option).</td>
</tr>
<tr>
<td>KRNUMMF</td>
<td>K_r region for matrix-fracture flows, for every grid block in the current input box (if using the Dual Porosity option).</td>
</tr>
<tr>
<td>LSLTWNUM</td>
<td>Low-salinity (oil-wet) saturation function regions.</td>
</tr>
<tr>
<td>LWSLTNUM</td>
<td>Low-salinity (water-wet) saturation function regions.</td>
</tr>
<tr>
<td>PENUM</td>
<td>Petro-elastic Model regions.</td>
</tr>
<tr>
<td>RESIDNUM</td>
<td>Relative permeability table numbers for Vertical Equilibrium residual flow option.</td>
</tr>
<tr>
<td>SURFNUM</td>
<td>Table numbers for the water-wet saturation functions and the mass of adsorbed surfactant versus fraction of oil-wet to water-wet curves, to use for each grid block, for the Surfactant option.</td>
</tr>
<tr>
<td>TRKPF</td>
<td>Region numbers for selecting which K(P) table to use for each partitioned tracer.</td>
</tr>
</tbody>
</table>

### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ODD3P option</td>
<td>DSTNUM</td>
<td>Grid-block tertiary hysteresis saturation table numbers</td>
</tr>
<tr>
<td></td>
<td>ALPHANUD</td>
<td>Defines transport coefficient table regions for drainage processes.</td>
</tr>
<tr>
<td>ODD3P option</td>
<td>ALPHANUI</td>
<td>Defines transport coefficient table regions for imbibition processes.</td>
</tr>
<tr>
<td></td>
<td>ALPHANUM</td>
<td>Defines transport coefficient table regions.</td>
</tr>
<tr>
<td></td>
<td>EOSNUM</td>
<td>Equation of state regions.</td>
</tr>
<tr>
<td></td>
<td>ISTNUM</td>
<td>Grid-block secondary hysteresis saturation table numbers</td>
</tr>
<tr>
<td></td>
<td>LGRCOPY</td>
<td>Copy global values to local cells before operations on cells within an LGR.</td>
</tr>
</tbody>
</table>
### SOLUTION section

The **SOLUTION** section contains sufficient data to define the initial state (pressure, saturations, compositions) of every grid block in the reservoir.

The keywords in the **SOLUTION** section may be specified in any order. All keywords must start in column 1. All characters up to column 8 are significant.

This data may take any one of the following forms:

#### Equilibration

Initial pressures and saturations are computed by ECLIPSE using data entered with the **EQUIL** keyword (for example the fluid contact depths).

#### Restart

The initial solution may be read from a Restart file created by an earlier run of ECLIPSE. The name of the Restart file is entered using the **RESTART** keyword.

#### Enumeration

You may specify the initial solution explicitly for every grid block.

See "Initializing the Study" in the **ECLIPSE Technical Description** for more information.
Equilibration - the EQUIL keyword

A line of EQUIL data must be specified for each equilibration region. (Cells may be assigned to equilibration regions using EQUNUM in the REGIONS section). If there is more than one equilibration region, it is important that they are non-communicating.

The EQUIL data specifies the initial pressure at a reference depth, the initial water-oil and gas-oil contact depths and the capillary pressures at these depths, and the equilibration options. For example:

```
EQUIL
9035  3600  9209  0  9035  0  1  1  20 /
```

This specifies:

- The pressure at datum depth of 9035 is 3600.
- Water oil contact ($P_{cow} = 0$) is at 9209 ft.
- Gas-oil contact ($P_{cog} = 0$) is at 9035 ft.
- 20 sub-intervals used in each cell for initial averaging.

When capillary pressure exists, an initial saturation distribution is found that matches this against the hydrostatic pressure gradient due to phase density differences. The 20 refers to the number of sub-intervals averaged to obtain the initial solution. The default is 1 sub-interval. One sub-interval should be used if an initial gas oil contact is present in a compositional run, or to obtain exact initial quiescence.

Composition variation with depth in the initial state may be specified in the black oil case using keywords such as RSVD (or PBVD), to specify $R_s$ or $P_{bub}$; and RVVD (or PDVD) to specify $R_v$ or $P_{dew}$. Either RSVD or PBVD is optional, to set $R_s$ or $P_{bub}$ versus depth. Either RVVD or PDVD is optional, to set $R_v$ or $P_{dew}$ versus depth.

In the compositional case, the compositions can be initialized using ZMFVD and COMPVD, described in the discussion of the PROPS section, or a constant composition may be entered with the ZI keyword in the PROPS section.

Enumeration

You may specify the initial solution for every grid block using the following keywords.

Note: The Enumeration option is not recommended for use in general.

Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PRESSURE</td>
<td>Cell pressures</td>
</tr>
<tr>
<td>SWAT</td>
<td>Water saturations</td>
</tr>
<tr>
<td>SGAS</td>
<td>Gas saturations</td>
</tr>
<tr>
<td>RS</td>
<td>Liquid gas-oil ratios</td>
</tr>
</tbody>
</table>
### ECLIPSE 100 only keywords
- **PRVD** Pressure versus depth variation.

### ECLIPSE 300 only keywords
- **SOIL** Oil saturations

In the compositional case, all of **PRESSURE**, **SWAT**, **SGAS**, **SOIL** may be used, and the initial composition is defined by the following keywords.
- **XMF** Liquid phase compositions
- **YMF** Vapor phase compositions

The **XMFVP** and **YMFVP** keywords, entered in the **PROPS** section, may be used as an alternative to **XMF** and **YMF** to define initial phase compositions.

In the thermal case the temperature must be specified.
- **TEMPI** Temperature

The **TEMPVD** keyword entered in the **PROPS** section may be used as an alternative to **TEMPI**.

As an alternative to entering these for the whole reservoir, the **ADD**, **COPY**, **EQUALS** and **MULTIPLY** keywords may be used to define values for **PRESSURE**, **SWAT**, **SGAS**, **RS**, **RV**.

See also the **GETSOL** and **GETDATA** keywords.

### Thermal enumeration with the NEI keyword

For the **THERMAL** option, the oil phase composition can be specified using the **NEI** keyword, instead of the **XMF** and **YMF** keywords. The simulator then finds a composition that is in thermodynamic equilibrium at the specified pressure and temperature and that gives the correct phase saturations. See “Initialization with NEI” in the **ECLIPSE Technical Description**.

### Flexible restarts

Flexible restarts may be specified using the **RESTART** keyword in place of the **EQUIL** keyword of a normal equilibration run. This loads up the initial solution written when Restart files have been requested by the **RPTRST** or **OUTSOL** keyword. A flexible restart only loads the initial solution - all other quantities such as transmissibilities and pore volumes are recalculated from the data supplied.

### Initial state output

Output of the initial solution to the Print file is controlled by the **RPTSOL** keyword. This is followed by a series of mnemonics, each specifying a solution quantity to be written. For example:
This requests output of the initial pressure and phase saturations. A large number of quantities can be output, and are listed under the RPTSOL keyword. It is only required that sufficient characters be entered to specify the required mnemonics uniquely.

**Analytic aquifers**

Carter-Tracy, Fetkovich, Constant Flux and Constant Head analytic aquifers may be specified with the following keywords (see also keyword AQUUDMS):

<table>
<thead>
<tr>
<th>Aquifer Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Carter-Tracy aquifers</td>
<td>AQUCT and AQUANCON / AQUANCONL (AQUUTAB in PROPS section)</td>
</tr>
<tr>
<td>Fetkovich aquifers</td>
<td>AQUFETP and AQUANCON/AQUANCONL, or AQUFET (ECLIPSE 100 only)</td>
</tr>
<tr>
<td>Constant flux aquifers</td>
<td>AQUFLUX and AQUANCON / AQUANCONL</td>
</tr>
<tr>
<td>Constant head gas aquifers</td>
<td>AQUCHGAS and AQUANCON / AQUANCONL</td>
</tr>
<tr>
<td>Constant head water aquifers</td>
<td>AQUCHWAT and AQUANCON / AQUANCONL</td>
</tr>
</tbody>
</table>

*Table 2.8: Analytical aquifer keywords*

For the multi-component water option (COMPW), water composition for AQUCHWAT, AQUCT, AQUFETP and AQUFLUX, AQSTREAM:

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>E100 only</td>
<td>AQUALIST</td>
<td>Aquifer lists</td>
</tr>
</tbody>
</table>

*Table 2.9: Aquifer lists*

See "Aquifer Modeling Facilities" in the *ECLIPSE Technical Description* for more information.

**Tracers, brine and API tracking**

The API option (RUNSPEC keyword API), the Brine option (RUNSPEC keyword BRINE), and the Tracer Tracking option (RUNSPEC keyword TRACERS) require the initial distribution to be provided by keywords as follows:

<table>
<thead>
<tr>
<th>Equilibration</th>
<th>API</th>
<th>Brine</th>
<th>Tracers</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>APIVD</td>
<td>SALTVD</td>
<td>TVDP</td>
</tr>
<tr>
<td></td>
<td>one table per equilibration region</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Enumeration</td>
<td>OILAPI</td>
<td>SALT</td>
<td>TBLK</td>
</tr>
<tr>
<td></td>
<td>for each grid block</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*Table 2.10: SOLUTION keywords for specifying initial distribution for API*

See "API Tracking", "Brine Tracking" and "Tracer Tracking" in the *ECLIPSE Technical Description* for further information.
Miscellaneous options

- **THPRES** specifies the threshold pressure (see RUNSPEC keyword EQLOPTS).

Initial fluid in place

ECLIPSE 300

An initial fluid in place report is always provided. In a compositional case, it is possible to set up a field separator, to define initial separator oil and gas, and used to calculate separator oil and gas in place throughout the run. (This is independent of the separator conditions in the well model).

The FIELDSEP keyword might take the following form:

```
FIELDSEP
1 80 815   /
2 60  14.7 /
```

This specifies a two stage separator, with the second stage representing the stock tank. The temperature and pressure is specified for each stage. The temperature is in degrees F or degrees C. The default coupling is from the liquid output of the first stage to the second. More complex configurations or more stages may be entered. If a FIELDSEP keyword is entered, this becomes the default separator conditions for wells in the SCHEDULE section.

A gas plant table may be used instead of a flash calculation for a separator stage (see item 7 of keyword FIELDSEP). In almost all gases, a single stage only is required. Gas plant tables are defined with keyword GPTABLE, GPTABLEN or GPTABLE3.

A recovery plant option, enabled by the RECOVERY keyword, may be used that takes all the gas exported from the field and processes this into NGLs and remaining gas. All other field quantities, such as GOR and gas-injection are reported prior to this processing. The aim of this facility is to model processing of the gas produced by the reservoir through facilities such as coolers, which can extract additional liquids from the stock tank gas.

The FIPSEP keyword allows different separators to be defined for different fluid-in-place regions. This would be of use if different separation processes were used in different reservoirs that are within a single simulation.

ECLIPSE 100

In the black oil case, an initial fluid-in-place report must be specified using the mnemonic in RPTSOL. The surface conditions are determined from the formation volume factors and surface density data entered in the PROPS section.

SOLUTION section keywords

A list, by function, of keywords that may occur in the SOLUTION section is given below, together with a brief description of their function. An alphabetic list follows. A more detailed description may be found by looking up the keyword in the manual.

Output

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optional</td>
<td>RPTSOL</td>
<td>Report arguments for SOLUTION data.</td>
</tr>
<tr>
<td>Optional</td>
<td>BOUNDARY</td>
<td>Defines section of grid for which output maps are required.</td>
</tr>
</tbody>
</table>
### Equilibration

#### Common keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required for equilibration.</td>
<td>EQUIL</td>
<td>Fluid contact depths and other equilibration parameters.</td>
</tr>
<tr>
<td>Optional for equilibration in live oil systems.</td>
<td>RSVD</td>
<td>Variation of solution GOR with depth.</td>
</tr>
<tr>
<td>Optional for equilibration in live oil systems.</td>
<td>PBVD</td>
<td>Variation of bubble point pressure with depth.</td>
</tr>
<tr>
<td>Optional for equilibration in wet gas systems.</td>
<td>RVVD</td>
<td>Variation of vapor OGR with depth.</td>
</tr>
<tr>
<td>Optional for equilibration in wet gas systems.</td>
<td>PDVD</td>
<td>Variation of dew point pressure with depth.</td>
</tr>
<tr>
<td>Required for equilibration when using Brine option.</td>
<td>SALTVD</td>
<td>Variation of salt concentration with depth.</td>
</tr>
<tr>
<td>Optional for initialization of tracer concentration.</td>
<td>TVDP</td>
<td>Variation of tracer concentration with depth.</td>
</tr>
<tr>
<td>Optional.</td>
<td>THPRES</td>
<td>Threshold pressures for flow between equilibration regions.</td>
</tr>
</tbody>
</table>

#### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required for equilibration when using API tracking.</td>
<td>APIVD</td>
<td>Variation of oil API with depth</td>
</tr>
</tbody>
</table>

#### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optional.</td>
<td>OLDEQUIL</td>
<td>Perform Equilibration using pre-2000A method</td>
</tr>
<tr>
<td>Optional, for the CO2SOL and GASSOL options.</td>
<td>.RSWVD</td>
<td>Aqueous concentration of dissolved gases versus depth</td>
</tr>
<tr>
<td>Thermal only</td>
<td>AMFVD</td>
<td>Aqueous mole fractions versus depth.</td>
</tr>
</tbody>
</table>
Restarting

The restart keywords are:

- **OUTSOL** Controls solution output for GRAF and flexible restarts. Optional
- **RESTART** Name of the restart file.
- **RPTRST** Report arguments for restart data. Optional

Enumeration

Common keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Either RS or PBUB required for enumeration in live oil systems.</td>
<td>PBUB</td>
<td>Bubble point pressures in every grid block.</td>
</tr>
<tr>
<td>Either RV or PDEW required for enumeration in wet gas systems.</td>
<td>PDEW</td>
<td>Dew point pressures in every grid block.</td>
</tr>
<tr>
<td>Either PRESSURE or PRVD is required for enumeration.</td>
<td>PRESSURE</td>
<td>Pressures in every grid-block</td>
</tr>
<tr>
<td>Either RS or PBUB required for enumeration in live oil systems.</td>
<td>RS</td>
<td>Solution gas-oil ratios in every grid block.</td>
</tr>
<tr>
<td>Either RV or PDEW required for enumeration in wet gas systems.</td>
<td>RV</td>
<td>Vapor oil-gas ratios in every grid block.</td>
</tr>
<tr>
<td>Required for enumeration when using Brine option.</td>
<td>SALT</td>
<td>Salt concentration values in every grid block.</td>
</tr>
<tr>
<td>Required for enumeration in two/three-phase systems with gas.</td>
<td>SGAS</td>
<td>Gas saturation in every grid-block.</td>
</tr>
<tr>
<td>Required for enumeration in two/three-phase systems with water</td>
<td>SWAT</td>
<td>Water saturation in every grid-block</td>
</tr>
<tr>
<td>Required for enumeration of tracer concentrations.</td>
<td>TBLK</td>
<td>Initializes concentration of tracers in every grid block</td>
</tr>
</tbody>
</table>

**ECLIPSE 100 only keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required for enumeration when using API tracking</td>
<td>OILAPI</td>
<td>Oil API gravity values in every grid block</td>
</tr>
<tr>
<td>Either PRESSURE or PRVD is required for enumeration.</td>
<td>PRVD</td>
<td>Pressure versus depth.</td>
</tr>
<tr>
<td></td>
<td>SFOAM</td>
<td>Initial Foam concentrations.</td>
</tr>
<tr>
<td></td>
<td>SPOLY</td>
<td>Initial Polymer concentrations.</td>
</tr>
<tr>
<td></td>
<td>SURF</td>
<td>Initial Surfactant concentrations</td>
</tr>
<tr>
<td>Restriction</td>
<td>ECLIPSE 100 only keyword</td>
<td>Description</td>
</tr>
<tr>
<td>---------------------------------------------------------------------------</td>
<td>--------------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Optional for systems with both live oil and wet gas.</td>
<td>VAPPARS</td>
<td>Parameters controlling vaporization of oil in under saturated gas</td>
</tr>
</tbody>
</table>

**ECLIPSE 300 only keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optional</td>
<td>AMF</td>
<td>Cell initial aqueous phase composition in GASWAT option.</td>
</tr>
<tr>
<td>Optional.</td>
<td>NEI</td>
<td>Specify compositions for non-equilibrium initialization.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Optional, for the CO2SOL and GASSOL options.</td>
<td>RSW</td>
<td>Aqueous concentration of dissolved gases in every grid block.</td>
</tr>
<tr>
<td>Either RV or PDEW required for enumeration in wet gas systems.</td>
<td>RV</td>
<td>Vapor oil-gas ratios in every grid block.</td>
</tr>
<tr>
<td>Solid option.</td>
<td>SMF</td>
<td>Cell initial solid composition.</td>
</tr>
<tr>
<td>Optional.</td>
<td>SOIL</td>
<td>Oil saturation in every grid-block.</td>
</tr>
<tr>
<td>SOR option.</td>
<td>SOILR</td>
<td>Cell residual oil saturation.</td>
</tr>
<tr>
<td>Solid option.</td>
<td>SSOLID</td>
<td>Solid saturation in every grid-block.</td>
</tr>
<tr>
<td>Optional.</td>
<td>XMF</td>
<td>Cell initial oil composition.</td>
</tr>
<tr>
<td>Optional.</td>
<td>YMF</td>
<td>Cell initial gas composition.</td>
</tr>
<tr>
<td>Optional.</td>
<td>ZMF</td>
<td>Cell initial total composition.</td>
</tr>
</tbody>
</table>

**Aquifer data**

**Common keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>AQANTRC</td>
<td>Initialize concentration of water phase tracers in each analytic aquifer.</td>
</tr>
<tr>
<td></td>
<td>AQUANCON</td>
<td>Connects an analytic aquifer, which was defined by AQUCT, AQUFETP, AQUFLUX, AUCHWAT and AUCHGAS, to one or more reservoir cells.</td>
</tr>
<tr>
<td></td>
<td>AQUANCONL</td>
<td>Connects an analytic aquifer which was defined by AQUCT, AQUFETP, AQUFLUX, AUCHWAT and AUCHGAS, to one or more reservoir cells located within local grid refinements</td>
</tr>
</tbody>
</table>
### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AQANNC</td>
<td>Specifies analytic aquifer connections explicitly.</td>
</tr>
<tr>
<td>AQUFET</td>
<td>Complete Fetkovich aquifer specifications.</td>
</tr>
<tr>
<td>AQUFLUX</td>
<td>Constant flux aquifer data specification.</td>
</tr>
<tr>
<td>AQUALIST</td>
<td>Aquifer lists.</td>
</tr>
<tr>
<td>RAINFALL</td>
<td>Specifies monthly flux rates for constant flux aquifers.</td>
</tr>
</tbody>
</table>

### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AQSTREAW</td>
<td>For the multi-component water option (COMPW), sets water composition for AUCHWAT, AUCT, AUFETP and AUFUX.</td>
</tr>
</tbody>
</table>

### Temperature option (ECLIPSE 100)

RTEMP or RTEMPA or RTEMPVD required when using the Temperature option.

- **RTEMP** Sets the initial reservoir temperature in each grid block.
- **RTEMPA** Sets the initial reservoir temperature in each grid block.
- **RTEMPVD** Sets the initial reservoir temperature versus depth.

### Other options

#### Common keywords

<table>
<thead>
<tr>
<th>Option</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coal Bed Methane</td>
<td>GASCONC</td>
<td>Initial coal gas concentration.</td>
</tr>
</tbody>
</table>
### Option | Keyword | Description
--- | --- | ---
Coal Bed Methane | GASSATC | Initial saturated coal gas concentration.

### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>Option</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coal Bed Methane</td>
<td>GCVD</td>
<td>Initial coal gas concentration with depth.</td>
</tr>
<tr>
<td>Gi (enumeration only)</td>
<td>GI</td>
<td>Initial Gi values.</td>
</tr>
<tr>
<td>Coal Bed Methane</td>
<td>SCVD</td>
<td>Initial coal solvent concentration with depth.</td>
</tr>
<tr>
<td>Solvent</td>
<td>SSOL</td>
<td>Initial solvent saturation.</td>
</tr>
</tbody>
</table>

### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>Option</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coal Bed Methane</td>
<td>GASADCO</td>
<td>Initial coal gas concentration for each component.</td>
</tr>
<tr>
<td>Coal Bed Methane</td>
<td>GASADEC</td>
<td>Initial saturated/equilibrium coal gas concentration for each component.</td>
</tr>
<tr>
<td>Coal Bed Methane</td>
<td>GASCCMP</td>
<td>Specify the component number for GASCONC and GASSATC.</td>
</tr>
<tr>
<td>THERMAL</td>
<td>TEMPI</td>
<td>Initial temperatures.</td>
</tr>
<tr>
<td>Coal Bed Methane</td>
<td>SORBFRAC</td>
<td>Scale initial adsorbed gas in the coal.</td>
</tr>
<tr>
<td>Coal Bed Methane</td>
<td>SORBRES</td>
<td>Specify the initial sorption pressure for the adsorbed gas in the coal.</td>
</tr>
</tbody>
</table>

### Operational keywords

#### Common keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optional.</td>
<td>ADD</td>
<td>Add value to an array.</td>
</tr>
<tr>
<td>Optional.</td>
<td>ADDREG</td>
<td>Add by FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>Optional.</td>
<td>BOX</td>
<td>Redefines the current input box.</td>
</tr>
<tr>
<td>Optional.</td>
<td>COPY</td>
<td>Copies data in the current box from one specified region to another.</td>
</tr>
<tr>
<td>Optional.</td>
<td>COPYREG</td>
<td>Copy by FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>Optional.</td>
<td>ENDBOX</td>
<td>Redefines the current input box to encompass the whole grid.</td>
</tr>
<tr>
<td>Optional.</td>
<td>ENDFIN</td>
<td>End input for local grid refinement.</td>
</tr>
<tr>
<td>Optional.</td>
<td>EQUALREG</td>
<td>Set by FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>Optional.</td>
<td>EQUALS</td>
<td>Set array to constant.</td>
</tr>
<tr>
<td>Optional.</td>
<td>IMPORT</td>
<td>Imports binary data generated by a grid pre-processor.</td>
</tr>
<tr>
<td>Optional.</td>
<td>MULTIPLY</td>
<td>Multiply array to constant.</td>
</tr>
<tr>
<td>Optional.</td>
<td>MULTIREG</td>
<td>Multiply array by FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td>Optional.</td>
<td>OPERATE</td>
<td>Performs arithmetic operations on arrays.</td>
</tr>
</tbody>
</table>
Restriction | Keyword | Description
--- | --- | ---
Optional. | OPERATOR | Performs arithmetic operations on arrays for a specific region of the grid (defined using the OPERNUM keyword in the GRID or REGIONS section).
Optional. | REFINE | Initiates data for a named local grid.

**ECLIPSE 100 only keywords**

- **GETDATA** Obtained named solution arrays from a restart of Initial file. Optional.
- **GETGLOB** Get global solution arrays from Restart file. Optional.

**ECLIPSE 300 only keywords**

- **GETSOL** Obtain initial solution arrays from a Restart file. Optional.

**Dynamic Regions Facility**

- **DYNAMICR** Defines regions based on when a set of conditions are satisfied. These regions may be static or allowed to change dynamically.
- **ENDDYN** Indicates the end of a DYNAMICR keyword sequence.

**Rivers (ECLIPSE 100)**

- **RIVERSYS** Defines a river system.

**Tuning (ECLIPSE 300)**

- **FCCRIT** Flash convergence criteria. Optional.

**ODD3P option (ECLIPSE 300)**

- **DBGODD3P** Diagnostic information for ODD3P option. Optional.

**Alphabetic list of keywords**

**Common keywords**

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>ADD</td>
<td>Add value to an array.</td>
</tr>
<tr>
<td></td>
<td>ADDREG</td>
<td>Add by FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td></td>
<td>AQANCONL</td>
<td>Connects an analytic aquifer which was defined by AQUCT, AQUFETP, AQUFLUX, AQUCHWAT and AQUCHGAS, to one or more reservoir cells located within local grid refinements.</td>
</tr>
<tr>
<td></td>
<td>AQANTRC</td>
<td>Initializes concentration of water phase tracers in each analytic aquifer.</td>
</tr>
<tr>
<td></td>
<td>AQUANCON</td>
<td>Connects an analytic aquifer which was defined by AQUCT, AQUFETP, AQUFLUX, AQUCHWAT and AQUCHGAS, to one or more reservoir cells.</td>
</tr>
<tr>
<td>Restriction</td>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>-------------</td>
<td>---------</td>
<td>-------------</td>
</tr>
<tr>
<td></td>
<td>AQUCHGAS</td>
<td>Constant head gas aquifer data specification.</td>
</tr>
<tr>
<td></td>
<td>AQUCHWAT</td>
<td>Constant head water aquifer data specification.</td>
</tr>
<tr>
<td></td>
<td>AQUCT</td>
<td>Carter-Tracy aquifer property data specification.</td>
</tr>
<tr>
<td></td>
<td>AQUFETP</td>
<td>Fetkovich aquifer property data specification.</td>
</tr>
<tr>
<td></td>
<td>AQUFLUX</td>
<td>Constant flux aquifer data specification.</td>
</tr>
<tr>
<td></td>
<td>BOUNDARY</td>
<td>Defines section of grid for which output maps are required.</td>
</tr>
<tr>
<td></td>
<td>BOX</td>
<td>Redefines the current box.</td>
</tr>
<tr>
<td></td>
<td>COPY</td>
<td>Copies data in the current box from one specified region to another.</td>
</tr>
<tr>
<td></td>
<td>COPYREG</td>
<td>Copy by FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td></td>
<td>DATUM</td>
<td>Defines datum depth for output of depth corrected pressures.</td>
</tr>
<tr>
<td></td>
<td>DATUMR</td>
<td>Defines datum depth for each FIP region.</td>
</tr>
<tr>
<td></td>
<td>DATUMRX</td>
<td>Defines datum depth for each FIP region in each FIP set.</td>
</tr>
<tr>
<td></td>
<td>DYNAMICR</td>
<td>Defines regions based on when a set of conditions are satisfied. These regions may be static or allowed to change dynamically.</td>
</tr>
<tr>
<td></td>
<td>ENDBOX</td>
<td>Redefines the current box to encompass the whole grid.</td>
</tr>
<tr>
<td></td>
<td>ENDDYN</td>
<td>Indicates the end of a DYNAMICR keyword sequence.</td>
</tr>
<tr>
<td></td>
<td>ENDFIN</td>
<td>End input for local grid refinement.</td>
</tr>
<tr>
<td></td>
<td>EQUALREG</td>
<td>Set by FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td></td>
<td>EQUALS</td>
<td>Set array to constant.</td>
</tr>
<tr>
<td></td>
<td>EQUIL</td>
<td>Fluid contact depths and other equilibration parameters.</td>
</tr>
<tr>
<td></td>
<td>FILEUNIT</td>
<td>Confirms the units of the data set.</td>
</tr>
<tr>
<td></td>
<td>GASCONC</td>
<td>Initial coal gas concentration.</td>
</tr>
<tr>
<td></td>
<td>GASSATC</td>
<td>Initial coal gas saturated concentration.</td>
</tr>
<tr>
<td></td>
<td>IMPORT</td>
<td>Imports binary data generated by a grid pre-processor.</td>
</tr>
<tr>
<td></td>
<td>MULTIPLY</td>
<td>Multiply array by constant.</td>
</tr>
<tr>
<td></td>
<td>MULTIREG</td>
<td>Multiply array by FLUXNUM, MULTNUM or OPERNUM region.</td>
</tr>
<tr>
<td></td>
<td>OPERATE</td>
<td>Performs arithmetic operations on arrays.</td>
</tr>
<tr>
<td></td>
<td>OPERATER</td>
<td>Performs arithmetic operations on arrays for a specific region of the grid (defined using the OPERNUM keyword in the GRID or REGIONS section).</td>
</tr>
<tr>
<td></td>
<td>PBUB</td>
<td>Bubble point pressures in every grid block.</td>
</tr>
<tr>
<td></td>
<td>PBVD</td>
<td>Variation of bubble point pressure with depth.</td>
</tr>
<tr>
<td></td>
<td>PDEW</td>
<td>Dew point pressures in every grid block.</td>
</tr>
<tr>
<td></td>
<td>PDVD</td>
<td>Variation of dew point pressure with depth.</td>
</tr>
<tr>
<td></td>
<td>PRESSURE</td>
<td>Pressures in every grid-block.</td>
</tr>
<tr>
<td></td>
<td>REFINE</td>
<td>Initiates data for a named local grid.</td>
</tr>
</tbody>
</table>
### Restriction | Keyword | Description
--- | --- | ---
RESTART | Name of the restart file.
RPTRST | Report arguments for restart data
RPTSOL | Report arguments for SOLUTION data.
RS | Solution gas-oil ratios in every grid-block.
RSVD | Variation of solution GOR with depth
RTEMPA | Sets the initial reservoir temperature in each grid block.
RTEMPVD | Sets the initial reservoir temperature versus depth.
RV | Vapor oil-gas ratios in every grid-block.
RVVD | Variation of vapor OGR with depth.
SALT | Salt concentrations in every grid block.
SALTVD | Variation of salt concentration with depth.
SGAS | Gas saturation in every grid-block.
SWAT | Water saturation in every grid-block.
TBLK | Initializes concentration of tracers in each grid cell.
THPRES | Threshold pressures for flow between equilibration regions.
TVDP | Variation of tracer concentration with depth.
VISDATES | Specify VISAGE geomechanics stress step dates.
VISOPTS | Specify VISAGE geomechanics workflow options.

## ECLIPSE 100 only keywords

| ECLIPSE 100 only keyword | Description |
--- | ---
APIVD | Variation of oil API with depth.
AQANNC | Specifies analytic aquifer connections explicitly.
AQUFET | Complete Fetkovich aquifer specifications.
GCVD | Initial coal gas concentration with depth.
GETDATA | Read named arrays from an ECLIPSE restart of Initial file.
GETGLOB | Get global solution arrays from Restart file.
GI | Initial Gi values.
OILAPI | Oil API gravity values in every grid-block.
PRVD | Pressure versus depth.
RIVERSYS | Defines a river system.
SCVD | Initial coal solvent concentration with depth.
SSOL | Initial solvent saturation.
VAPPARS | Parameters controlling vaporization of oil in under saturated gas.
# ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AMF</td>
<td></td>
<td>Specifies cell initial aqueous composition explicitly.</td>
</tr>
<tr>
<td>AQSTREAM</td>
<td></td>
<td>Sets gas composition for use with AUCHGAS aquifer and, for the GASWAT and CO2STORE options, water composition for use with AUCHWAT, AUCT, AQUFETP and AQUFLUX.</td>
</tr>
<tr>
<td>AQSTREAW</td>
<td></td>
<td>For the multi-component water option (COMPW), sets water composition for use with AUCHWAT, AUCT, AQUFETP and AQUFLUX.</td>
</tr>
<tr>
<td>DBGODD3P</td>
<td></td>
<td>Debug output for ODD3P option.</td>
</tr>
<tr>
<td>FCCRIT</td>
<td></td>
<td>Flash convergence criteria.</td>
</tr>
<tr>
<td>FIELDSEP</td>
<td></td>
<td>Introduces a field separator</td>
</tr>
<tr>
<td>FIPSEP</td>
<td></td>
<td>Introduces a separator to be used in a fluid-in-place region.</td>
</tr>
<tr>
<td>GASADCO</td>
<td></td>
<td>Initial coal gas concentration.</td>
</tr>
<tr>
<td>GASADEC</td>
<td></td>
<td>Initial coal gas equilibrium/saturated concentration.</td>
</tr>
<tr>
<td>GETSOL</td>
<td></td>
<td>Obtain initial solution arrays from a Restart file.</td>
</tr>
<tr>
<td>GPTABLE</td>
<td></td>
<td>Tables of gas plant recovery factors.</td>
</tr>
<tr>
<td>GPTABLEN</td>
<td></td>
<td>Gas plant table with NGL recoveries.</td>
</tr>
<tr>
<td>GPTABLE3</td>
<td></td>
<td>Gas plant table with oil, NGL and gas recoveries.</td>
</tr>
<tr>
<td>NEI</td>
<td></td>
<td>Specify compositions for non-equilibrium initialization</td>
</tr>
<tr>
<td>OLDEQUIL</td>
<td></td>
<td>Perform Equilibration using pre-2000A method.</td>
</tr>
<tr>
<td>OUTSOL</td>
<td></td>
<td>Controls solution output for GRAF and flexible restarts</td>
</tr>
<tr>
<td>RECOVERY</td>
<td></td>
<td>Recovery plant table.</td>
</tr>
<tr>
<td>SOR option.</td>
<td>ROMF</td>
<td>Cell residual oil composition.</td>
</tr>
<tr>
<td></td>
<td>RSW</td>
<td>Aqueous concentration of CO₂ in every grid block.</td>
</tr>
<tr>
<td></td>
<td>RSWVD</td>
<td>Aqueous concentration of CO₂ versus depth.</td>
</tr>
<tr>
<td></td>
<td>SMF</td>
<td>Cell initial solid composition.</td>
</tr>
<tr>
<td></td>
<td>SOIL</td>
<td>Oil saturation in every grid-block.</td>
</tr>
<tr>
<td>SOR option.</td>
<td>SOILR</td>
<td>Cell residual oil saturation.</td>
</tr>
<tr>
<td></td>
<td>SSOLID</td>
<td>Solid saturation in every grid-block.</td>
</tr>
<tr>
<td></td>
<td>TEMPI</td>
<td>Initial temperatures.</td>
</tr>
<tr>
<td>water component model</td>
<td>WI</td>
<td>Initial water composition</td>
</tr>
<tr>
<td>water component model</td>
<td>WMF</td>
<td>Cell initial water composition</td>
</tr>
</tbody>
</table>
### SUMMARY section

The `SUMMARY` section specifies a number of variables that are to be written to Summary files after each timestep of the simulation. The graphics post-processor may be used to display the variation of variables in the Summary files with time and with each other. If there is no `SUMMARY` section, ECLIPSE does not create any Summary files.

The keywords that may be specified in the `SUMMARY` section are shown in the following tables. All are optional, and no significance attaches to the order in which they are specified. All keywords must start in column 1. All characters up to column 8 are significant.

#### Keyword data format

Summary vector keywords can have one of three formats:

- keyword with no data;
- keyword with a list of data, terminated by a slash (/). Sometimes the list can be defaulted (left blank);
- keyword followed by a set of records, each record terminated by a slash (/). A blank record is used to terminate the keyword

Examples of these three formats are:

```plaintext
FPR

WBHP 'WELL1' 'WELL2' 'WELL3' /

BPR
  1 1 1 /
  1 1 2 /
  1 1 3 /

/```

**Note:** When inputting a user defined name (for example well name, well list name, group name) it is important to enclose the name in quotes for example `PROD1`. Failure to do so may lead to unexpected results.

For more information on the `SUMMARY` file format for different keyword groups, see the information starting with "Format of SUMMARY keywords beginning with A".

For more information on `SUMMARY` keywords with a component index see, "Format of SUMMARY keywords with a component index".

---

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>water component model</td>
<td>WMFVVD</td>
<td>Initial water composition as a function of depth</td>
</tr>
<tr>
<td></td>
<td>XMF</td>
<td>Cell initial oil composition.</td>
</tr>
<tr>
<td></td>
<td>YMF</td>
<td>Cell initial gas composition</td>
</tr>
<tr>
<td></td>
<td>ZMF</td>
<td>Cell initial total composition.</td>
</tr>
</tbody>
</table>
**Format of SUMMARY keywords beginning with A**

Keywords that begin with the letter 'AA' are followed by a list of analytic aquifer numbers terminated by a slash (/). Keywords that begin with the letter 'AN' are followed by a list of numerical aquifer numbers terminated by a slash (/).

For example,

```
AAQR
  1 2 /
```

specifies that analytic aquifers 1 and 2 are to be stored in the summary files. If a null list of aquifers is specified, data is stored in the Summary file for all analytic (liquid or gas depending on the keyword) or numerical aquifers.

**ECLIPSE 300**

Keywords that begin with the letter 'AAC' or 'ANC' are followed by a list of pairs composed of a respectively analytic or numerical aquifer number and a component number. Each pair must begin on a fresh line and must be terminated by a slash (/). The list is terminated by a line that begins with a slash. The component number can be defaulted, in which case all components allowed in the given aquifer will be stored in the summary file.

For example,

```
AACMR
  1 2 /
  / 
```

specifies that component 2 for analytic aquifer 1 is to be stored in the summary file. If a null list is specified, data is stored in the summary file for all analytic or numerical aquifers and all components allowed in them.

The keyword **ALL** is an exception to this rule. There are no data associated with it.

Output for both water aquifer (**AQUCT**, **AQUFLUX**, **AQUCHWAT**, **AQUFETP**, and **AQUFET** (ECLIPSE 100 only)), and gas aquifer (**AQUCHGAS**) rates and totals are done using these mnemonics. Must ensure that correct aquifer number is specified for this output.

**ECLIPSE 100**

Keywords which begin with the letters 'AL' are followed by a list of analytic aquifer list names terminated by a slash (/). For example,

```
ALQR
  'AQLIST1'
  'AQLIST2'
  / 
```

specifies that the aquifer list influx rate for aquifer lists **AQLIST1** and **AQLIST2** are to be stored in the summary file.

**Format of SUMMARY keywords beginning with B**

Keywords that begin with the letter 'b' are followed by a list of sets of block coordinates. Each set of coordinates must begin on a fresh line and must be terminated by a slash (/). The list is terminated by a line that begins with a slash.

For example,

```
BOSAT
  8 1 14 /
```
Keywords that begin with a B and end with a _X are used to report linear interpolated values and thus require a different format. The input XYZ coordinates are validated and updated against the input cell block IJK coordinates, which can be defaulted in simple grids to allow ECLIPSE automatic calculation. When IJK coordinates are updated, a message is output. When IJK coordinates cannot be found given input XYZ coordinates, a WARNING message is issued and a block centered value is reported for cell IJK.

A reference height can be defined in place of global value specified in HYDRHEAD. The reference height is used only for pressure and hydraulic head keywords BPR_X, BHD_X and BHDF_X. Each set of coordinates must begin on a fresh line and must be terminated by a slash (/). The list is terminated by a line that begins with a slash.

For example,

```
BPR_X
-- I J K Cell   X Y Z Coordinate   Reference-Height   GroupName
  8 1 14     238.5 -78.5  -2.5   1*                 'AP'/
  1* 1* 1*     122.5 -46.5  -7.5   100.0              'AP'/
/```

If the discretized matrix block option for dual porosity is active (keyword NMATRIX in the RUNSPEC section), the solution inside the matrix cells can be obtained by appending the required ring number to the end of the following summary keywords: BOSAT, BWSAT, BGSAT, BPR, BRS and BRV. Keywords without a ring number appended refer to the outermost matrix ring adjacent to the fracture.

For example,

```
BPR3
3 3 6 /
4 3 6 /
/
```

Block keywords that require a component index, such as BMLSC, BXMF, BYMF, BGMCO and BGMCG are described in Format of SUMMARY keywords with a component index.

### Format of SUMMARY keywords beginning with C

Keywords that begin with the letter C must be followed by a list of well-reservoir connections (the well name and the three block coordinates). Each connection must be specified on a fresh line and terminated by a slash (/). The three grid block coordinates may be defaulted for a well, which will cause the appropriate quantity to be output for all connections in the well. The list of well connections is terminated by a line beginning with a slash (/).

**Note:** If the connection coordinates are defaulted, as for well PROD3 below, ECLIPSE reserves a number of summary vectors equal to the maximum number of connections per well (NCWMAX, set in RUNSPEC section keyword WELLDIMS). If this number is much greater than the actual number of connections in well PROD3 then defaulting the connections can become quite wasteful.

For example,

```
COFR
'PROD1' 10 3 7 /
'PROD1' 10 3 8 /
'PROD2' 4 2 5 /
```
The well name may be defaulted by using the character ‘*’. If both the well name and the connection coordinates are defaulted (as in the example below), then the requested summary quantities are output for all well connections in the global model. If the well name is defaulted but the connection coordinates are provided, then the requested summary quantities will be output for all wells with a connection in the specified block. Please note that well name templates are currently not allowed.

Keywords beginning with the letter ‘C’ and ending with the letter ‘L’ output the corresponding connection quantity totaled over all the connections that belong to the same lumped completion (see keyword COMPLUMP). Any connection within the lumped completion can be specified to define the completion. Note that the well name may not be defaulted for a lumped completion quantity.

For the conductive faults option the summary keywords (see table 2.72) should be followed by a list of conductive fault names and should be terminated by a slash (/). Defaulting the fault name list results in output for all conductive faults. For example,

```
CFOSAT 'FLT1' 'FLT2' /  -- Oil saturation in faults 'FLT1' and 'FLT2'
CFPRES /                 -- Pressure in all faults
```

Completion keywords that require a component index, such as CZMF, CKFR and CKFT are described in "Format of SUMMARY keywords with a component index".

Field keywords that require a component index, such as FXMF, FYMF, FZMF, FCMPR, FCMO, FCGMI and FCGMM are described in "Format of SUMMARY keywords with a component index". All other keywords that begin with the letter 'F' have no associated data.

Keywords that begin with 'G' must be followed by a list group names enclosed in quotes. The list is terminated by a slash (/).

For example,

```
GWCT 'PLAT1' 'PLAT2' 'PLAT3' /  -- Water cut for the three named groups of wells
```

If a null list is specified, data is stored for all groups.

Group keywords that require a component index are described in "Format of SUMMARY keywords with a component index".
Format of SUMMARY keywords beginning with LB

A local block (LB) keyword should be followed by a list of LGR names and local block coordinates. Each block must be specified on a fresh line, terminated by a slash (/), and the list of blocks is terminated by a line containing only a slash (/).

Note, in ECLIPSE 100 global block data associated with LGR host cells can be accessed via the B*SUMMARY keywords. For ECLIPSE 300 block data associated with LGR host cells is not available and data should either be requested for unrefined cells using the B* keywords, or refined cells using the LB* keywords.

There is a known limitation for tracer block quantities as the tracer name should be appended to the LB* keyword. As the total number of characters in the SUMMARY keyword cannot exceed 8, the tracer name may need to be reduced in consequence.

Local block interpolation keyword begin with a LB and ends with a _X are used to report linear interpolated values and thus require a different format. The local LGR grid name should be included. The input XYZ coordinates are validated and updated against the input local cell block IJK coordinates, which can be defaulted in simple grids to allow ECLIPSE automatic calculation. When local IJK coordinates are updated, a message is output. When local IJK coordinates cannot be found given input XYZ coordinates, a WARNING message is issued and value is reported as block centered value for cell IJK.

A reference height can be defined in place of global value specified in HYDRHEAD. The reference height is used only for pressure and hydraulic head keywords LBPR_X, LBHD_X and LBHDF_X. Each set of coordinates must begin on a fresh line and must be terminated by a slash (/). The list is terminated by a line that begins with a slash.

Format of SUMMARY keywords beginning with LC

A local well connection (LC) keyword should be followed by a list of LGR names, well names and the local block coordinates. Each connection must be specified on a fresh line and terminated by a slash (/). The list of connections is terminated by a line containing only a slash (/).

If the connection coordinates are defaulted for a well (as in well PROD2 in the example below), the corresponding quantity is output for all connections in the well (within the specified LGR).
The well name may be defaulted by using the character ‘*’ (as in the examples below). If both the well name and the local connection coordinates are defaulted, then the requested summary quantities are output for all well connections in the specified LGR. If the well name is defaulted but the connection coordinates are provided, then the requested summary quantities will be output for all wells with a connection in the specified local block.

Note: If the connection coordinates are defaulted, ECLIPSE 100 reserves a number of Summary vectors equal to the maximum number of connections per well (NCWMAX, set in RUNSPEC section keyword WELLDIMS). If this number is much greater than the actual number of connections in the local grid well, defaulting the connection coordinates can become quite wasteful.

Format of SUMMARY keywords beginning with LW

Local well (LW) keywords should be followed by a list of LGR names and well names. Each line of data should be terminated with a slash (/) and the set of records must end with a blank line, containing only a slash (/).

If the well name contains wildcard characters then the template will be matched during the simulation period. See "Well name and well list template matching" in the ECLIPSE Technical Description. Alternatively, if the well name is defaulted then all wells within the named LGR are stored in the summary file. For example:

```
LWOPR
-- LGR Name  Well Name
  'LGR1'   '*'
  'LGR2'   '*'
  'LGR3'   
```

For LGR amalgamations, any LGR within the amalgamation may be used to locate the well. The entered LGR is used in the summary output.

ECLIPSE 100
The keywords with a separator stage index are not available for local wells.

Format of SUMMARY keywords beginning with R

Inter-region keywords

Keywords that begin with the letter 'R' and have the letters 'FT' at characters 3 and 4 or at characters 4 and 5 (for example RGFTL and RNLFT) are used to specify that the total flow of a particular phase between particular pairs of fluid-in-place regions is to be stored in the summary files. Keywords that begin with the letter 'R' and have the letters 'FR' at characters 3 and 4 or at characters 4 and 5 (for example RGFR and RNLFR) are used to specify that the rate flow of a particular phase between particular pairs of fluid-in-place regions is to be stored in the summary files. An exception to this pattern is RORFR which is a region summary keyword. Each pair of region numbers must be specified on a fresh line, and terminated by a
slash (/). The list is terminated by a line beginning with a slash (/). Such keywords are described as “inter-
region flow”.

For example,

```
RWFT
  1 2 /
  1 3 /
  2 3 /
/`
```

Additionally, keywords that start with RKFT or RKFR accept a component number appended to the fluid-
in-place region pair. This component number can be defaulted, in which case all components will be stored
in the summary file.

For example,

```
RKFT
  1 2 4 /
  1 3 4 /
  2 3 /
/`
```

Inter-region keywords can be obtained for the additional set of fluid-in-place regions by compounding them
with the first 3 characters of the name of the set of fluid-in-place regions. Summary quantities having less
than 5 characters should be extended to 5 characters with an underscore before adding the 3-character
region set name.

```
ROFR_FST     Inter-region oil flow rate from region 1 to 2
  1 2 /
/        of the set of regions named FST

ROFR+FST     Positive contribution of Inter-region oil flow rate
  1 2 /
/        from region 1 to 2 of the set of regions named FST
```

**Region keywords**

Other keywords which begin with the letter 'R' (for example RWIP) are followed by a list of fluid-in-place
region numbers which is terminated by a single slash (/). The region numbers are not organized in pairs.

For example,

```
ROIP
  5 6
  8 /
```

specifies that the oil in place in regions 5, 6 and 8 is to be stored in the summary files. If a null list of
regions is specified, data is stored in the Summary file for all regions.

For example:

```
RGIPG
/`
```

If additional sets of fluid-in-place regions are being used (defined with keyword FIP in the REGIONS
section), then the SUMMARY keywords beginning with the letter R are available to request output from these
additional sets of regions. In this case, the region quantity keyword should be compounded with the first 3
characters of the name of the set of fluid-in-place regions specified with the FIP keyword. SUMMARY keywords with less than 5 characters (for example ROIP) should be extended to five characters with underscores before appending the three-character region set name. For example:

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ROIPLFST</td>
<td>for the oil-in-place in the liquid phase for region 2</td>
</tr>
<tr>
<td>ROIP_SND</td>
<td>for the total oil-in-place in all the regions in the set named SND</td>
</tr>
<tr>
<td>RPR__THR</td>
<td>for the pressure in all the regions in the set named THR</td>
</tr>
</tbody>
</table>

At present the keywords associated with tracers within a region (that is those beginning with RT) are not available for the additional sets of fluid-in-place regions.

Some keywords which begin with the letter ‘R’ and which are not inter-region flow keywords accept a component number as noted in their description (for instance RCOM, RCWM, RCGMI and RCGMM). Such summary keywords are specified using two items, the first corresponding to the region number and the second corresponding to the component index. The component number can be defaulted, in which case data for all components will be output in the summary file. For example:

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RCOM</td>
<td>Molar amount of component 2 dissolved in oil, within region 1 of standard FIP set.</td>
</tr>
<tr>
<td>RCGMIFST</td>
<td>Molar amount of all components trapped in gas, within region 2 of the set of region named FST</td>
</tr>
</tbody>
</table>

**Format of SUMMARY keywords beginning with S**

Keywords that begin with ‘S’ request data for a segment of a multisegment well or river and must be followed by a list records consisting of well names and segment numbers, with each record terminated by a slash (/). The final record should be followed by a further slash. For example:

<table>
<thead>
<tr>
<th>Record</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SOFR</td>
<td></td>
</tr>
<tr>
<td>'WELL1' 2 /</td>
<td></td>
</tr>
<tr>
<td>'WELL1' 3 /</td>
<td></td>
</tr>
<tr>
<td>'WELL2' /</td>
<td></td>
</tr>
</tbody>
</table>

If the segment number is omitted from a record then data for all segments for that well will be stored.

**Note:** If the segment number is defaulted, as for well WELL2 above, ECLIPSE will reserve a number of summary vectors equal to the maximum number of segments per well (NSEGMX, set in the RUNSPEC section keyword **WSEGDIMS**). If this number is much greater than the actual number of segments in well WELL2 then defaulting the segment numbers can become quite wasteful. This also applies to SUMMARY keywords that begin with ‘S’ applied to river reaches.

The well name may be defaulted. Data for all wells and all segments can be generated by using the appropriate keyword followed by an empty record containing a single slash.

If the well name is omitted, a wildcard used or a well list used then no data will be stored for that record.

Segment keywords that require a component index, such as SKFR and SCWGFR are described in "Format of SUMMARY keywords with a component index".

The SUMMARY keywords **STEP_TYPE**, **SEPARATE** and **SUM_THIN** are exceptions to the above and take no arguments.
**Format of SUMMARY keywords beginning with W**

Keywords that begin with 'W' must be followed by a list of well names enclosed in quotes. The list is terminated by a slash (/). For example:

```
WBHP
  'WELL1' 'WELL2' 'WELL3'
```

If a null list is specified, data is stored for all wells. For example,

```
WBHP

```

A well name or well list template containing wildcard characters may be supplied to a keyword beginning with the letter 'W'. The template will then be matched during the course of the simulation, see "Well name and well list template matching" in the ECLIPSE Technical Description.

Keywords beginning with the letter 'W' and ending with the letter 'L' are for lumped completions (with the exception of WMCTL) and are an alternative form of the equivalent keyword which begins with the letter 'C'. They should be followed by a well name and the index of the required lumped completion. They output the corresponding connection quantity totaled over all the connections that belong to the same lumped completion (see keyword COMPLUMP). For example,

```
WOPRL
  'P1' 2
```

The mnemonic name written to the Summary file for these well keywords has the lumped completion index appended after additional underscore characters (for example, WOPRL__2). If the completion index is less than 10, two underscore characters are used. If it is less than 100 and bigger than 9, one underscore character is used. If it is less than 1000 and bigger than 99, no underscore character is used.

**ECLIPSE 300**

Well keywords that require a component index, such as WXMF and WYMF are described in "Format of SUMMARY keywords with a component index".

**Format of SUMMARY keywords that end in H**

The keywords that end with the letter H (for example WOPRH, WWPRH, WGPRH, WWCTH, WGORH, WOGRH, WWGRH) refer to the observed production/injection history of history matching wells specified in keywords WCONHIST and WCONINJH as long as the description mentions WCONHIST or WCONINJH. These quantities are included to enable you to compare, for example, the calculated water cut WWCT with the observed water cut WWCTH. They are ignored if applied to wells that are not history matching wells. Group and field keywords ending in H (for example GOPRH, FWPRH) refer to the sum of their subordinate well flows, with the flows of all history matching wells replaced with their historical rates specified in WCONHIST and WCONINJH. Thus, for example, FWCT can be compared with FWCTH to determine how well the field water cut matches its observed value.

**Format of SUMMARY keywords with a component index**

Keywords that provide information at a component level must have the component number appended to any other data associated with the keyword. For group type keywords a line of data must contain a single group name followed by a component number terminated by a slash (/). For well type keywords a line of data can either contain a single well name or a well name template containing wildcard characters, followed by a component number and terminated by a slash (/). If a well name template is supplied then it will be
matched during the simulation period. See "Well name and well list template matching" in the ECLIPSE Technical Description.

When rates and totals are being reported for groups or wells at a component level, they are positive for production and negative for injection.

For example,

```
WXMF
'PROD1' 7 /
'PROD2' / -- list all components
```

or:

```
GXMF
'PLAT1' 7 /
'PLAT1' 8 /
'PLAT2' / -- list all components
```

or, to get the 7th, 8th and 9th component values of BXMF at grid block (21,7,13) and all components of grid block (21,7,14):

```
BXMF
21 7 13 7 /
21 7 13 8 /
21 7 13 9 /
21 7 14 / -- list all components
```

or (note that this format is different from that used for keywords beginning with B or W):

```
FZMF
1 /  
FZMF
2 /
FZMF
/ -- list all components
```

or:

```
CZMF
'PROD1' 7 7 1 7 /
'PROD1' 7 7 1 8 /
'PROD2' 7 7 1 / -- list all components
```

or:

```
SKFR
'PROD1' 7 1 /
'PROD1' 8 2 /
'PROD2' 3 / -- list all components
```

Note that keywords FREAC and FREAT allow multiple reaction numbers:

```
FREAC
1 2/ -- Reaction rates summed over all cells for reactions 1 and 2
```
For keywords that start with B, C, S, F, G or W the component number can be defaulted, in which case all components will be stored in the summary file.

Keywords that begin with A and accept a component number are described in "Format of SUMMARY keywords beginning with A".

Keywords that begin with R and accept a component number are described in "Format of SUMMARY keywords beginning with R".

The mnemonic name written to the Summary file has the component index appended after an additional underscore character (for example, \texttt{WXMF\_7}). In rare cases the underscore will be omitted to ensure the composite name does not exceed 8 characters (for example, although the molar rate of component 3 in the gas from the 2nd separator stage of a well is written as \texttt{WCGMR2\_3}, the molar rate of component 15 has to be written as \texttt{WCGMR215}).

Water components

For the multi-component water option, the water component index starts at \( \mathcal{N}_c \) + 1 and ends at \( \mathcal{N}_c + \mathcal{N}_w \) where \( \mathcal{N}_c \) is the number of components given by \texttt{COMPS} and \( \mathcal{N}_w \) is the number of water components given by the \texttt{COMPW} keyword. The water component indices can be used for the summary mnemonics described in "SUMMARY section keywords" in the \textit{ECLIPSE Technical Description} manual.

Keywords for special options

The keywords that refer specifically to network pressures or the Polymer, Brine, Solvent and Coal bed methane options should only be used when these options are active. These are ECLIPSE Special Options, and you are referred to the relevant sections of the \textit{ECLIPSE Technical Description} manual for further details.

Maximum number of keywords

The total number of quantities that are written to the Summary file must not exceed \texttt{NSUMMX} set in \texttt{RUNSPEC} keyword \texttt{SMRYDIMS}.

\textbf{Note:} For example, \texttt{WOPR} applied to four individual wells counts as four separate quantities towards this total.

It is quite simple to construct a context-independent \texttt{SUMMARY} section that can be \texttt{INCLUDE} d in any run, and which provides sufficient data for many requirements.

Example SUMMARY section

Here is an example of a context independent \texttt{SUMMARY} section:

```plaintext
SUMMARY  ==================================================================================
-- Field oil prod rate.  
-- Cumulative oil prod for field and for every well.
FOPR
FOPT
WOPT /
-- Field wat inj rate. Cumulative wat inj for field and for every well.
FWIR
FWIT
WWIT
```
Summary file names

A fresh Summary file is created at each report time. They have names like BASE.S0011 which is the Summary file containing data for all timesteps in the period between reports 10 and 11 in a run whose filename root is ‘BASE’.

ECLIPSE also creates a file called BASE.SMSPEC, which contains a specification of the data in the Summary files. The naming convention can vary slightly according to the make of computer.

These files may be formatted (for transfer between computers of different makes, or just for readability), or unformatted (the default). The files are formatted if keyword FMTOUT is present in the RUNSPEC section. Formatted files have slightly different names, to distinguish them from unformatted files. The file names are of the form BASE.A0011 and BASE.FSMSPEC.

Well and group flow quantities

The well and group oil flow quantities are given in the table below. Notes for this table:

1. See “Format of SUMMARY keywords with a component index”.

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Completion</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>Field</td>
<td>Group</td>
<td>Well</td>
<td>COFR</td>
<td>COFRL</td>
<td>Oil Flow Rate (+ve=prod -ve=inj)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>Well</td>
<td>WOFRL</td>
<td></td>
<td>Oil Flow Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>Well</td>
<td>COPRL</td>
<td></td>
<td>Oil Flow Rate (as above)</td>
</tr>
<tr>
<td>E100</td>
<td>E300</td>
<td>Field</td>
<td>Group</td>
<td>Well</td>
<td>Connection</td>
<td>Completion</td>
<td>Information</td>
</tr>
<tr>
<td>------</td>
<td>------</td>
<td>-------</td>
<td>-------</td>
<td>------</td>
<td>------------</td>
<td>------------</td>
<td>-------------</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>WOPRL</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Oil Flow Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>COFRF</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Free Oil Flow Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>COFRS</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Solution oil flow rate (vaporized oil)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>COFRU</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Sum of connection oil flow rates upstream of, and including, this connection</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPR</td>
<td>GOPR</td>
<td>WOPR</td>
<td>COPR</td>
<td></td>
<td>Oil Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPRA</td>
<td>GOPRA</td>
<td>WOPRA</td>
<td></td>
<td></td>
<td>Oil Production Rate above GOC</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPRB</td>
<td>GOPRB</td>
<td>WOPRB</td>
<td></td>
<td></td>
<td>Oil Production Rate below GOC</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPTA</td>
<td>GOPTA</td>
<td>WOPTA</td>
<td></td>
<td></td>
<td>Oil Production Total above GOC</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPTB</td>
<td>GOPTB</td>
<td>WOPTB</td>
<td></td>
<td></td>
<td>Oil Production Total below GOC</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPR1</td>
<td>GOPR1</td>
<td>WOPR1</td>
<td></td>
<td></td>
<td>Oil Production Rate above GOC (molar method)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPR2</td>
<td>GOPR2</td>
<td>WOPR2</td>
<td></td>
<td></td>
<td>Oil Production Rate below GOC (molar method)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPT1</td>
<td>GOPT1</td>
<td>WOPT1</td>
<td></td>
<td></td>
<td>Oil Production Total above GOC (molar method)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPT2</td>
<td>GOPT2</td>
<td>WOPT2</td>
<td></td>
<td></td>
<td>Oil Production Total below GOC (molar method)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOMR</td>
<td>GOMR</td>
<td>WOMR</td>
<td></td>
<td></td>
<td>Oil Mass Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOMT</td>
<td>GOMT</td>
<td>WOMT</td>
<td></td>
<td></td>
<td>Oil Mass Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FODN</td>
<td>GODN</td>
<td>WODN</td>
<td></td>
<td></td>
<td>Oil Density at Surface Conditions</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPRH</td>
<td>GOPRH</td>
<td>WOPRH</td>
<td></td>
<td></td>
<td>Oil Production Rate History (\texttt{WCONHIST})</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPRT</td>
<td>GOPRT</td>
<td>WOPRT</td>
<td></td>
<td></td>
<td>Oil Production Rate Target/Limit</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GOPRL</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Oil Production Rate Target/Limit (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPRF</td>
<td>GOPRF</td>
<td>WOPRF</td>
<td></td>
<td></td>
<td>Free Oil Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPRS</td>
<td>GOPRS</td>
<td>WOPRS</td>
<td></td>
<td></td>
<td>Solution Oil Production Rate (vaporized oil)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPT</td>
<td>GOPT</td>
<td>WOPT</td>
<td>COPT</td>
<td>COPTL</td>
<td>Oil Production Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>WOPTL</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Oil Production Total (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPTH</td>
<td>GOPTH</td>
<td>WOPTH</td>
<td></td>
<td></td>
<td>Oil Production Total History (\texttt{WCONHIST})</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPTF</td>
<td>GOPTF</td>
<td>WOPTF</td>
<td>COPTF</td>
<td></td>
<td>Free Oil Production Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPTS</td>
<td>GOPTS</td>
<td>WOPTS</td>
<td>COPTS</td>
<td></td>
<td>Solution Oil Production Total (vaporized oil)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOIR</td>
<td>GOIR</td>
<td>WOIR</td>
<td></td>
<td></td>
<td>Oil Injection Rate</td>
</tr>
</tbody>
</table>
### Table 2.11: Well and group oil flows

The well and group water flow quantities are given in the table below. Notes for this table:

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Completion</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>FOIRH</td>
<td>GOIRH</td>
<td>WOIRH</td>
<td></td>
<td></td>
<td>Oil Injection Rate History ((\text{WCONINJH}))</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOIRT</td>
<td>GOIRT</td>
<td>WOIRT</td>
<td></td>
<td></td>
<td>Oil Injection Rate Target/Limit</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>GOIRL</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Oil Injection Rate Target/Limit (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOIT</td>
<td>GOIT</td>
<td>WOIT</td>
<td></td>
<td></td>
<td>Oil Injection Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>COIT</td>
<td></td>
<td>COITL</td>
<td></td>
<td></td>
<td>Oil Injection Total (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>WOITL</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Oil Injection Total (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOITH</td>
<td>GOITH</td>
<td>WOITH</td>
<td></td>
<td></td>
<td>Oil Injection Total History ((\text{WCONINJH}))</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPP</td>
<td>GOPP</td>
<td>WOPP</td>
<td></td>
<td></td>
<td>Oil Potential Production rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>COPP</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Oil Potential Production rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOPP2</td>
<td>GOPP2</td>
<td>WOPP2</td>
<td></td>
<td></td>
<td>Oil Potential Production rate (including values calculated on shut/stopped wells)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FOPI</td>
<td>GOPI</td>
<td>WOPI</td>
<td>COPI</td>
<td></td>
<td>Oil Potential Injection rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>COPI</td>
<td></td>
<td>WOPI</td>
<td></td>
<td></td>
<td>Oil Potential Injection rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GOPGR</td>
<td>WOPGR</td>
<td></td>
<td></td>
<td></td>
<td>Oil Production Guide Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>GOIGR</td>
<td>WOIGR</td>
<td></td>
<td></td>
<td></td>
<td>Oil Injection Guide Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOVPR</td>
<td>GOVPR</td>
<td>WOVPR</td>
<td></td>
<td></td>
<td>Oil Voidage Production Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FOVPT</td>
<td>GOVPT</td>
<td>WOVPT</td>
<td></td>
<td></td>
<td>Oil Voidage Production Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOVIR</td>
<td>GOVIR</td>
<td>WOVIR</td>
<td></td>
<td></td>
<td>Oil Voidage Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FOVIT</td>
<td>GOVIT</td>
<td>WOVT</td>
<td></td>
<td></td>
<td>Oil Voidage Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOnPR</td>
<td>GOnPR</td>
<td>WOnPR</td>
<td></td>
<td></td>
<td>(n)th separator stage oil rate ((n = 1) to 99). Note that the oil volume rate is calculated at the temperature and pressure of the separator stage and not at stock tank conditions</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FOnPT</td>
<td>GOnPT</td>
<td>WOnPT</td>
<td></td>
<td></td>
<td>(n)th separator stage oil total ((n = 1) to 99)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FEOR</td>
<td>GEOR</td>
<td></td>
<td></td>
<td></td>
<td>Export Oil Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FEOT</td>
<td>GEOT</td>
<td></td>
<td></td>
<td></td>
<td>Export Oil Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FEMF</td>
<td>GEOMF</td>
<td></td>
<td></td>
<td></td>
<td>Export Oil Mole Fraction. See Note 1.</td>
</tr>
</tbody>
</table>
1. See “Format of SUMMARY keywords with a component index”.

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Completion</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CWFR</td>
<td>CWFRL</td>
<td>Water Flow Rate (+ve=prod - ve=inj)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WWFR</td>
<td>Water Flow Rate (as above)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WWFRL</td>
<td>Water Flow Rate (as above)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WWFR</td>
<td>Water Flow Rate (as above)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WWFRL</td>
<td>Water Flow Rate (as above)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Sum of connection water flow rates upstream of, and including, this connection</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FWFR</td>
<td>GWPR</td>
<td>WWFR</td>
<td></td>
<td>CWPR</td>
<td>Water Production Rate</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FWMR</td>
<td>GWMR</td>
<td>WWMR</td>
<td></td>
<td></td>
<td>Water Mass Rate</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FWMT</td>
<td>GWMT</td>
<td>WWMT</td>
<td></td>
<td></td>
<td>Water Mass Total</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FWPRH</td>
<td>GWPRH</td>
<td>WWPRH</td>
<td></td>
<td></td>
<td>Water Production Rate History (WCONHIST)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FWPT</td>
<td>GWPT</td>
<td>WWPT</td>
<td></td>
<td>CWPTL</td>
<td>Water Production Total</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WWPTL</td>
<td>Water Production Total (as above)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FWPTH</td>
<td>GWPTH</td>
<td>WWPTH</td>
<td></td>
<td></td>
<td>Water Production Total History (WCONHIST)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FWIR</td>
<td>GWIR</td>
<td>WWIR</td>
<td></td>
<td>CWIRL</td>
<td>Water Injection Rate</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WWIRL</td>
<td>Water Injection Rate (as above)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FWIRH</td>
<td>GWIRH</td>
<td>WWIRH</td>
<td></td>
<td></td>
<td>Water Injection Rate History (WCON1NJJH)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FWIRT</td>
<td>GWIRT</td>
<td>WWIRT</td>
<td></td>
<td></td>
<td>Water Injection Rate Target/Limit</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>GWIRT</td>
<td>Water Injection Rate Target/Limit (as above)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FWIT</td>
<td>GWIT</td>
<td>WWIT</td>
<td></td>
<td>CWITL</td>
<td>Water Injection Total</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WWITL</td>
<td>Water Injection Total (as above)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FWITH</td>
<td>GWITH</td>
<td>WWITH</td>
<td></td>
<td></td>
<td>Water Injection Total History (WCON1NJJH)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FWPP</td>
<td>GWPP</td>
<td>WWPP</td>
<td></td>
<td>CWPP</td>
<td>Water Potential Production rate</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WWPP</td>
<td>Water Potential Production rate (as above)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FWPP2</td>
<td>GWPP2</td>
<td>WWPP2</td>
<td></td>
<td></td>
<td>Water Potential Production rate (including values calculated on shut/stopped wells)</td>
</tr>
</tbody>
</table>
Table 2.12: Well and group water flows

The well and group gas flow options are given in the table below. Notes for this table:

1. See “Format of SUMMARY keywords with a component index”
2. See OPTIONS3 keyword item 108 for alternative reporting of this quantity during the surface gas production balancing group control mode.

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Completion</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>FWPI</td>
<td>GWPI</td>
<td>WWPI</td>
<td></td>
<td></td>
<td>Water Potential Injection rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CWPI</td>
<td></td>
<td>Water Potential Injection rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td>WWPI</td>
<td></td>
<td></td>
<td></td>
<td>Water Potential Injection rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FWPI2</td>
<td>GWPI2</td>
<td>WWPI2</td>
<td></td>
<td></td>
<td>Water Potential Injection rate (including values calculated on shut/stopped wells)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td>WWPI2</td>
<td></td>
<td></td>
<td></td>
<td>Water Potential Injection rate (including values calculated on shut/stopped wells) (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GWPIR</td>
<td>WWPIR</td>
<td></td>
<td></td>
<td></td>
<td>Water Production Guide Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GWIGR</td>
<td>WWIGR</td>
<td></td>
<td></td>
<td></td>
<td>Water Injection Guide Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FWVPR</td>
<td>GWVPR</td>
<td>WWVPR</td>
<td></td>
<td></td>
<td>Water Voidage Production Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FWVPT</td>
<td>GWVPT</td>
<td>WWVPT</td>
<td></td>
<td></td>
<td>Water Voidage Production Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FWVIR</td>
<td>GWVIR</td>
<td>WWVIR</td>
<td></td>
<td></td>
<td>Water Voidage Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FWVIT</td>
<td>GWVIT</td>
<td>WWVIT</td>
<td></td>
<td></td>
<td>Water Voidage Injection Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FWPIR</td>
<td>GWPIR</td>
<td>WWPIR</td>
<td></td>
<td></td>
<td>Ratio of produced water to injected water expressed as a percentage</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FWMPR</td>
<td>GWMPR</td>
<td>WWMPR</td>
<td></td>
<td></td>
<td>Water component molar production rate. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FWMPR</td>
<td>GWMPR</td>
<td>WWMPR</td>
<td></td>
<td></td>
<td>Water component molar production total. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FWMPR</td>
<td>GWMPR</td>
<td>WWMPR</td>
<td></td>
<td></td>
<td>Water component molar injection rate. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FWMPR</td>
<td>GWMPR</td>
<td>WWMPR</td>
<td></td>
<td></td>
<td>Water component molar injection total. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FWMPR</td>
<td>GWMPR</td>
<td>WWMPR</td>
<td></td>
<td></td>
<td>Water component molar injection rate. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FWMPR</td>
<td>GWMPR</td>
<td>WWMPR</td>
<td></td>
<td></td>
<td>Water component molar injection total. See Note 1.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Completion</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>CGFR</td>
<td>CGFRL</td>
<td>Gas Flow Rate (+ve=prod -ve=inj)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WGFRL</td>
<td>Gas Flow Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CGPRL</td>
<td>Gas Flow Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WGPRL</td>
<td>Gas Flow Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CGFRF</td>
<td></td>
<td>Free Gas Flow Rate</td>
</tr>
<tr>
<td>E100</td>
<td>E300</td>
<td>Field</td>
<td>Group</td>
<td>Well</td>
<td>Connection</td>
<td>Completion</td>
<td>Information</td>
</tr>
<tr>
<td>------</td>
<td>------</td>
<td>-------</td>
<td>-------</td>
<td>------</td>
<td>------------</td>
<td>------------</td>
<td>-------------</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CGFRS</td>
<td></td>
<td>Solution Gas Flow Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CGFRU</td>
<td></td>
<td>Sum of connection gas flow rates upstream of, and including, this connection</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FGPR</td>
<td>GGPR</td>
<td>WGPR</td>
<td>CGPR</td>
<td></td>
<td>Gas Production Rate Note: In the presence of lift gas (see the VFPPROD keyword), the produced gas reported does not include produced lift gas, only produced formation gas.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FGPR1</td>
<td>GGPR1</td>
<td>Gas Production Rate above GOC</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FGPR2</td>
<td>GGPR2</td>
<td>Gas Production Rate below GOC</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FGPTA</td>
<td>GGPTA</td>
<td>Gas Production Total above GOC</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FGPTB</td>
<td>GGPTB</td>
<td>Gas Production Total below GOC</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FGPR1</td>
<td>GGPR1</td>
<td>Gas Production Rate above GOC (molar method)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FGPR2</td>
<td>GGPR2</td>
<td>Gas Production Rate below GOC (molar method)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FGPT1</td>
<td>GGPT1</td>
<td>Gas Production Total above GOC (molar method)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FGPT2</td>
<td>GGPT2</td>
<td>Gas Production Total below GOC (molar method)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FGMR</td>
<td>GGMR</td>
<td>Gas Mass Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FGMT</td>
<td>GGMT</td>
<td>Gas Mass Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FGDN</td>
<td>GGDN</td>
<td>Gas Density at Surface Conditions</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FGPRH</td>
<td>GGPRH</td>
<td>WGPR</td>
<td>FGPRH</td>
<td></td>
<td>Gas Production Rate History (WCONHIST)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FGPT</td>
<td>GGPT</td>
<td>WGPT</td>
<td>FGPTL</td>
<td></td>
<td>Gas Production Rate Target/ Limit</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>FGPRF</td>
<td></td>
<td>Gas Production Rate Target/ Limit (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FGPRS</td>
<td></td>
<td>Free Gas Production Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FGPRS</td>
<td></td>
<td>Solution Gas Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>FGPT</td>
<td></td>
<td>Gas Production Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>FGPTL</td>
<td></td>
<td>Gas Production Total (as above)</td>
</tr>
<tr>
<td>E100</td>
<td>E300</td>
<td>Field</td>
<td>Group</td>
<td>Well</td>
<td>Connection</td>
<td>Completion</td>
<td>Information</td>
</tr>
<tr>
<td>------</td>
<td>------</td>
<td>-------</td>
<td>-------</td>
<td>-------</td>
<td>------------</td>
<td>-----------</td>
<td>-------------------------------------------------</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FGPTH</td>
<td>GGPTH</td>
<td>WGPTH</td>
<td></td>
<td></td>
<td>Gas Production Total History (WCONHIST)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FGPTF</td>
<td>GGPTF</td>
<td>WGPTF</td>
<td>CGPTF</td>
<td></td>
<td>Free Gas Production Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FGPTS</td>
<td>GGPTS</td>
<td>WGPTS</td>
<td>CGPTS</td>
<td></td>
<td>Solution Gas Production Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FGIR</td>
<td>GGIR</td>
<td>WGIR</td>
<td>CGIR</td>
<td>CGIRL</td>
<td>Gas Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WGIIRL</td>
<td>Gas Injection Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FGIRT</td>
<td>GGIRT</td>
<td>WGIIR</td>
<td></td>
<td></td>
<td>Gas Injection Rate Target/Limit</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>GGIRL</td>
<td>Gas Injection Rate Target/Limit (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FGIT</td>
<td>GGIT</td>
<td>WGIT</td>
<td>CGIT</td>
<td>CGITL</td>
<td>Gas Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WGIITL</td>
<td>Gas Injection Total (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FGITH</td>
<td>GGITH</td>
<td>WGIIT</td>
<td></td>
<td></td>
<td>Gas Injection Total History (WCONINJH)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FGPP</td>
<td>GGPP</td>
<td>WGPP</td>
<td></td>
<td></td>
<td>Gas Potential Production rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CGPP</td>
<td>Gas Potential Production rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FGPP2</td>
<td>GGPP2</td>
<td>WGP2</td>
<td></td>
<td></td>
<td>Gas Potential Production rate (including values calculated on shut/stopped wells)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FGPPS</td>
<td>GGPPS</td>
<td>WGPSS</td>
<td></td>
<td></td>
<td>Solution (dissolved) Gas Potential Production rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FGPPS2</td>
<td>GGPPS2</td>
<td>WGPSS2</td>
<td></td>
<td></td>
<td>Solution (dissolved) Gas Potential Production rate (including values calculated on shut/stopped wells)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FGPPF</td>
<td>GGPPF</td>
<td>WGPFF</td>
<td></td>
<td></td>
<td>Free Gas Potential Production rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FGPPF2</td>
<td>GGPPF2</td>
<td>WGPFF2</td>
<td></td>
<td></td>
<td>Free Gas Potential Production rate (including values calculated on shut/stopped wells)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FGPI</td>
<td>GGPI</td>
<td>WGPI</td>
<td></td>
<td></td>
<td>Gas Potential Injection rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CGPI</td>
<td>Gas Potential Injection rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WGIIP</td>
<td>Gas Potential Injection rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FGPI2</td>
<td>GGPI2</td>
<td>WGPI2</td>
<td></td>
<td></td>
<td>Gas Potential Injection rate (including values calculated on shut/stopped wells)</td>
</tr>
<tr>
<td>E100</td>
<td>E300</td>
<td>Field</td>
<td>Group</td>
<td>Well</td>
<td>Connection</td>
<td>Completion</td>
<td>Information</td>
</tr>
<tr>
<td>------</td>
<td>------</td>
<td>-------</td>
<td>-------</td>
<td>------</td>
<td>------------</td>
<td>------------</td>
<td>-------------</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Gas Potential Injection rate (including values calculated on shut/stopped wells) (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td>GGPGR</td>
<td></td>
<td></td>
<td></td>
<td>Gas Production Guide Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td>GGIGR</td>
<td></td>
<td></td>
<td></td>
<td>Gas Injection Guide Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td>FSGR</td>
<td>GSGR</td>
<td></td>
<td></td>
<td>Sales Gas Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td>FGSR</td>
<td>GGSR</td>
<td></td>
<td></td>
<td>Sales Gas Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td>FSGT</td>
<td>GSGT</td>
<td></td>
<td></td>
<td>Sales Gas Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td>FGST</td>
<td>GGST</td>
<td></td>
<td></td>
<td>Sales Gas Total (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FSGR</td>
<td>GSGR</td>
<td></td>
<td></td>
<td>Sales Gas Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FGSR</td>
<td>GGSR</td>
<td></td>
<td></td>
<td>Sales Gas Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FSGT</td>
<td>GSGT</td>
<td></td>
<td></td>
<td>Sales Gas Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FGST</td>
<td>GGST</td>
<td></td>
<td></td>
<td>Sales Gas Total (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FSGR</td>
<td>GSGR</td>
<td></td>
<td></td>
<td>Sales Gas Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FGSR</td>
<td>GGSR</td>
<td></td>
<td></td>
<td>Sales Gas Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FSGT</td>
<td>GSGT</td>
<td></td>
<td></td>
<td>Sales Gas Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FGST</td>
<td>GGST</td>
<td></td>
<td></td>
<td>Sales Gas Total (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FSGR</td>
<td>GSGR</td>
<td></td>
<td></td>
<td>Sales Gas Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FGSR</td>
<td>GGSR</td>
<td></td>
<td></td>
<td>Sales Gas Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FSGT</td>
<td>GSGT</td>
<td></td>
<td></td>
<td>Sales Gas Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FGST</td>
<td>GGST</td>
<td></td>
<td></td>
<td>Sales Gas Total (as above)</td>
</tr>
</tbody>
</table>

**Gas Lift Injection Rate** (Assumes ALQ = gas lift injection rate. Well rates are multiplied by their efficiency factors when summing up group and field rates.)

| x    |      |       | FGLIR | GGLIR | WGLIR |            | Gas Lift Injection Rate (Assumes ALQ = gas lift injection rate. Well rates are multiplied by their efficiency factors when summing up group and field rates.) |
| x    |      |       |       |       |       |            |            |
| x    |      |       |       |       |       |            |            |
| x    |      |       |       |       |       |            |            |

**Wet Gas Production Rate**

| x    |      |       | FWGPR | GWGPR | WWGPR |            | Wet Gas Production Rate |
| x    |      |       | FWGPT | GWGPT | WWGPT |            | Wet Gas Production Total |
| x    |      |       | FWGPRH | GWGPRH | WWGPRH |            | Wet Gas Production Rate History (**WCONHIST**) |
| x    |      |       | FWGIR | GWGIR | WWGIR |            | Wet Gas Injection Rate |
| x    |      |       | FWGIT | GWGIT | WWGIT |            | Wet Gas Injection Total |
| x    |      |       | FEGR  | GEGR  |       |            | Export Gas Rate. See Note 2. |
| x    |      |       | FEGT  | GEGT  |       |            | Export Gas Total. See Note 2. |
Table 2.13: Well and group gas flows

The well and group liquid flow options are:

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Completion</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>CLFR</td>
<td>CLFR</td>
<td>Liquid Flow Rate (+ve=prod - ve=inj)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WLFRL</td>
<td>WLFRL</td>
<td>Liquid Flow Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FLPR</td>
<td>GLPR</td>
<td>WLPR</td>
<td></td>
<td>WLFRL</td>
<td>Liquid Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FLPRH</td>
<td>GLPRH</td>
<td>WLP RH</td>
<td></td>
<td>WLFRL</td>
<td>Liquid Production Rate History (WCONHIST)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FLPT</td>
<td>GLPT</td>
<td>WLPT</td>
<td>CLPT</td>
<td>WLFPT</td>
<td>Liquid Production Rate Target/ Limit</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td>GLPRL</td>
<td></td>
<td></td>
<td>WLFPT</td>
<td>Liquid Production Rate Target/ Limit (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FLPT</td>
<td>GLPT</td>
<td>WLPT</td>
<td>CLPT</td>
<td>WLFPT</td>
<td>Liquid Production Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WLP TL</td>
<td>WLFPT</td>
<td>Liquid Production Total (as above)</td>
</tr>
</tbody>
</table>
Table 2.14: Well and group liquid flows

The well and group linearly combined flow options are:

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Completion</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>FLPTH</td>
<td>GLPTH</td>
<td>WLPHT</td>
<td></td>
<td></td>
<td>Liquid Production Total History (WCONHIST)</td>
</tr>
</tbody>
</table>

Table 2.15: Well and group linearly combined flows

The well and group reservoir volume flow options are:

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Completion</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>FJPR</td>
<td>GJPR</td>
<td>WJPR</td>
<td>CVFR</td>
<td>CVFRL</td>
<td>Reservoir (Res) Volume Flow Rate (+ve=prod -ve=inj)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FJPRH</td>
<td>GJPRH</td>
<td>WJPRH</td>
<td>CVFRL</td>
<td></td>
<td>Res Volume Flow Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FJPR</td>
<td>GJPR</td>
<td>WJPR</td>
<td>CVPR</td>
<td></td>
<td>Res Volume Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FJPRH</td>
<td>GJPRH</td>
<td>WJPRH</td>
<td>CVPR</td>
<td></td>
<td>Res Volume Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FJPR</td>
<td>GJPR</td>
<td>WJPR</td>
<td>WVPR</td>
<td>CVIRL</td>
<td>Res Volume Injection Flow Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FJPRH</td>
<td>GJPRH</td>
<td>WJPRH</td>
<td>WVIRL</td>
<td></td>
<td>Res Volume Injection Flow Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FVPRT</td>
<td>GVPT</td>
<td>WVPT</td>
<td>CVPT</td>
<td></td>
<td>Res Volume Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FVPRT</td>
<td>GVPT</td>
<td>WVPT</td>
<td>CVPT</td>
<td></td>
<td>Res Volume Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GVPRL</td>
<td>CVPR</td>
<td></td>
<td></td>
<td></td>
<td>Res Volume Production Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GVPRL</td>
<td>CVPR</td>
<td></td>
<td></td>
<td></td>
<td>Res Volume Production Rate (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FVPT</td>
<td>GVPT</td>
<td>WVPT</td>
<td>CVPTL</td>
<td></td>
<td>Res Volume Production Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FVPRT</td>
<td>GVPT</td>
<td>WVPT</td>
<td>CVPTL</td>
<td></td>
<td>Res Volume Production Total (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GVPGR</td>
<td>WVPGR</td>
<td></td>
<td></td>
<td></td>
<td>Res Volume Production Guide Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FVIR</td>
<td>GVIR</td>
<td>WVIR</td>
<td>CVIR</td>
<td></td>
<td>Res Volume Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FVIRT</td>
<td>GVIRT</td>
<td>WVIRT</td>
<td>CVIR</td>
<td></td>
<td>Res Volume Injection Rate (as above)</td>
</tr>
</tbody>
</table>
Table 2.16: Well and group reservoir volume flows

The group export flow options are given in the table below. Notes for this table:

1. See OPTIONS3 keyword item 108 for alternative reporting of this quantity during the surface gas production balancing group control mode.

2. See “Format of SUMMARY keywords with a component index”

Table 2.17: Group export flows (ECLIPSE 300)

The well and group calorific value flow options are:

Table 2.18: Well and group calorific value flows (ECLIPSE 300)

The well and group production ratio options are:

Table 2.19: Well and group production ratio options (ECLIPSE 300)
Table 2.19: Well and group production ratios

The well pressure and PI options are given in the table below. Notes for this table:

1. No units are given in FIELD, as noted in “The productivity index” in the ECLIPSE Technical Description

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Completion</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>WGORL</td>
<td>WGORL</td>
<td>Gas-Oil Ratio (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FGORH</td>
<td>GGORH</td>
<td>x</td>
<td>WGORL</td>
<td>FGORH</td>
<td>Gas-Oil Ratio History (WCONHIST)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOGR</td>
<td>GOGR</td>
<td>x</td>
<td>WGORL</td>
<td>FOGR</td>
<td>Oil-Gas Ratio</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FOGRH</td>
<td>GOGRH</td>
<td>x</td>
<td>WGORL</td>
<td>FOGRH</td>
<td>Oil-Gas Ratio History (WCONHIST)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FWGR</td>
<td>GWGR</td>
<td>x</td>
<td>WWRL</td>
<td>FWGR</td>
<td>Water-Gas Ratio</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>WWRL</td>
<td>WWRL</td>
<td>Water-Gas Ratio (as above)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FWGRH</td>
<td>GWGRH</td>
<td>x</td>
<td>WWRL</td>
<td>FWGRH</td>
<td>Water-Gas Ratio History (WCONHIST)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FGLR</td>
<td>GGLR</td>
<td>x</td>
<td>WGLRL</td>
<td>FGLR</td>
<td>Gas-Liquid Ratio</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>x</td>
<td>WGLRL</td>
<td>GGLR</td>
<td>Gas-Liquid Ratio</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FGLRH</td>
<td>GGLRH</td>
<td>x</td>
<td>WGLRL</td>
<td>FGLRH</td>
<td>Gas-Liquid Ratio History (WCONHIST)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>x</td>
<td>WBGLR</td>
<td></td>
<td>Bottom hole Gas-Liquid Ratio</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Well</th>
<th>Connection</th>
<th>Completion</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>CPR</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>CPRL</td>
<td>Average Connection Pressure in lumped completion</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>WBHP</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>CPRL</td>
<td>Average Connection Pressure in lumped completion</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>WBHPH</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>WBHPH</td>
<td>Bottom Hole Pressure History (WCONHIST, WCONINJH)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>WTHP</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>WTHP</td>
<td>Tubing Head Pressure History (WCONHIST, WCONINJH)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>WPI</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>CPI</td>
<td>Productivity Index of well’s preferred phase. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>WPIO</td>
<td>Oil phase PI</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>WPIG</td>
<td>Gas phase PI</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>WPIW</td>
<td>Water phase PI</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>WPIL</td>
<td>Liquid phase PI</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>CTFAC</td>
<td>Connection Transmissibility Factor</td>
</tr>
</tbody>
</table>
### Table 2.20: Well pressures and PIs

The well and group control options are:

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>CDBF</td>
<td>Blocking factor for generalized pseudo-pressure method</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>WBP</td>
<td>One-point Pressure Average (see keywords WPAVE and WWAVE)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>WBP4</td>
<td>Four-point Pressure Average (see keywords WPAVE and WWAVE)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>WBP5</td>
<td>Five-point Pressure Average (see keywords WPAVE and WWAVE)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>WBP9</td>
<td>Nine-point Pressure Average (see keywords WPAVE and WWAVE)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>WPI1</td>
<td>Productivity Index based on the value of WBP. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>WPI4</td>
<td>Productivity Index based on the value of WBP4. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>WPI5</td>
<td>Productivity Index based on the value of WBP5. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>WPI9</td>
<td>Productivity Index based on the value of WBP9. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WHD</td>
<td>Hydraulic head in well based on the reference depth given in HYDRHEAD and the well’s reference depth. Requires Water Services license.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WHDF</td>
<td>Hydraulic head in well based on the reference depth given in HYDRHEAD and the well’s reference depth calculated at freshwater conditions. Requires Water Services license.</td>
</tr>
</tbody>
</table>

Well State Indicator:

1.0 = PROD (Producer),
2.0 = INJ (Injector),
3.0 = SHUT (Shut for reasons other than priority control),
4.0 = STOP (Stopped for reasons other than priority control),
5.0 = PSHUT (Priority Shut),
6.0 = PSTOP (Priority Stop).

The output is rendered as mnemonics in the .RSM file.
## Mode of Control:

- **0.0** = SHUT/STOP,
- **1.0** = ORAT,
- **2.0** = WRAT,
- **3.0** = GRAT,
- **4.0** = LRAT,
- **5.0** = RESV,
- **6.0** = THP,
- **7.0** = BHP,
- **9.0** = CRAT,
- **11.0** = GOR Penalty,
- **12.0** = Drawdown,
- **13.0** = NGL,
- **30.0** = Availability,
- **31.0** = REIN,
- **32.0** = TMRA,
- **33.0** = WGRA,
- **34.0** = CVAL,

- **-1.0** = well under group control (ECLIPSE 300),
- **-ve** if rate target was set by group (ECLIPSE 100),

### Thermal Rates:

- **41.0** = STRA (Thermal: Steam Rate),
- **42.0** = SATP (Thermal: Saturated Pressure),
- **43.0** = SATT (Thermal: Saturated Temperature).

The output is rendered as mnemonics in the .RSM file.
<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>FMCTP</td>
<td>GMCTP</td>
<td></td>
<td>Mode of Control for group Production:</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.0 = NONE,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1.0 = ORAT,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2.0 = WRAT,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>3.0 = GRAT,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>4.0 = LRAT,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>5.0 = RESV,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>6.0 = PRBL,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>7.0 = ENERGY,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>8.0 = WGRA,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>9.0 = CRAT (E100), 9.0 = CVAL (E300),</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>10.0=PBGS,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>11.0=PBWS,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>12.0 = CRAT (E300),</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-ve if rate target was set by higher group.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>The output is rendered as mnemonics in the .RSM file.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>x</th>
<th>x</th>
<th>FMCTW</th>
<th>GMCTW</th>
<th></th>
<th>Mode of Control for group Water Injection:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.0 = NONE,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1.0 = RATE,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2.0 = RESV,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>3.0 = REIN,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>4.0 = VREP,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>8.0 = WGRA,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>9.0 = Availability,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-ve if rate target was set by higher group.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>The output is rendered as mnemonics in the .RSM file.</td>
</tr>
</tbody>
</table>
Mode of Control for group Gas Injection:
0.0 = NONE,
1.0 = RATE,
2.0 = RESV,
3.0 = REIN,
4.0 = VREP,
8.0 = WGRA,
9.0 = Availability,
-ve if rate target was set by higher group.
The output is rendered as mnemonics in the .RSM file.

Table 2.21: Well and group control modes

The group well count options are:

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>FMCTG</td>
<td>GMCTG</td>
<td></td>
<td>Mode of Control for group Gas Injection:</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.0 = NONE,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>1.0 = RATE,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2.0 = RESV,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>3.0 = REIN,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>4.0 = VREP,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>8.0 = WGRA,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>9.0 = Availability,</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-ve if rate target was set by higher group.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>The output is rendered as mnemonics in the .RSM file.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>FMWPT</td>
<td>GMWPT</td>
<td></td>
<td>Total number of production wells</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWPR</td>
<td>GMWPR</td>
<td></td>
<td>Number of production wells currently flowing</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWPA</td>
<td>GMWPA</td>
<td></td>
<td>Number of abandoned production wells</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWPU</td>
<td>GMWPU</td>
<td></td>
<td>Number of unused production wells</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWPG</td>
<td>GMWPG</td>
<td></td>
<td>Number of producers on group control</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWPO</td>
<td>GMWPO</td>
<td></td>
<td>Number of producers controlled by own oil rate limit</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWPS</td>
<td>GMWPS</td>
<td></td>
<td>Number of producers on own surface rate limit control</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWPV</td>
<td>GMWPV</td>
<td></td>
<td>Number of producers on own reservoir volume rate limit control</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWPP</td>
<td>GMWPP</td>
<td></td>
<td>Number of producers on pressure control</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWPL</td>
<td>GMWPL</td>
<td></td>
<td>Number of producers using artificial lift (with ALQ &gt; 0.0)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWIT</td>
<td>GMWIT</td>
<td></td>
<td>Total number of injection wells</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWIN</td>
<td>GMWIN</td>
<td></td>
<td>Number of injection wells currently flowing</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWIA</td>
<td>GMWIA</td>
<td></td>
<td>Number of abandoned injection wells</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWIU</td>
<td>GMWIU</td>
<td></td>
<td>Number of unused injection wells</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWIG</td>
<td>GMWIG</td>
<td></td>
<td>Number of injectors on group control</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWIS</td>
<td>GMWIS</td>
<td></td>
<td>Number of injectors on own surface rate limit control</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWIV</td>
<td>GMWIV</td>
<td></td>
<td>Number of injectors on own reservoir volume rate limit control</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWIP</td>
<td>GMWIP</td>
<td></td>
<td>Number of injectors on pressure control</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>WMCON</td>
<td></td>
<td></td>
<td>The number of connections capable of flowing in the well</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWDR</td>
<td>GMWDR</td>
<td></td>
<td>Number of drilling events this timestep</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWDT</td>
<td>GMWDT</td>
<td></td>
<td>Number of drilling events in total</td>
</tr>
</tbody>
</table>
Table 2.22: Group well counts

The well and group energy rate options are:

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>FMWNO</td>
<td>GMWNO</td>
<td></td>
<td>Number of workover events this timestep</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWWT</td>
<td>GMWWT</td>
<td></td>
<td>Number of workover events in total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>GMWL</td>
<td></td>
<td>Number of wells grouped together in a well list</td>
</tr>
</tbody>
</table>

Number of workover events this timestep

Number of workover events in total

Number of wells grouped together in a well list (either static or dynamic)
### Table 2.23: Well and group energy rates

The solid production totals mnemonics are:

<table>
<thead>
<tr>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>FSMR</td>
<td>GSMR</td>
<td>WSMR</td>
<td>Solid mass production rate</td>
</tr>
<tr>
<td>x</td>
<td>FSMT</td>
<td>GSMTP</td>
<td>WSMTP</td>
<td>Solid mass production total</td>
</tr>
<tr>
<td>x</td>
<td>FSVR</td>
<td>GSVR</td>
<td>WSVR</td>
<td>Solid volume production rate</td>
</tr>
<tr>
<td>x</td>
<td>FSVT</td>
<td>GSVT</td>
<td>WSVT</td>
<td>Solid volume production total</td>
</tr>
</tbody>
</table>

### Table 2.24: Solid production totals (ECLIPSE 300 Solid option)

#### Miscellaneous well and group quantities

The miscellaneous well and group quantity options are given in the table below. Notes for this table:

1. See “Format of SUMMARY keywords with a component index”.

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>CDSM</td>
<td>Current mass of scale deposited</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>CDSML</td>
<td>Current mass of scale deposited per unit perforation length</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>CDSF</td>
<td>PI multiplicative factor due to scale damage</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>GEFF</td>
<td>Efficiency Factor</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WFF</td>
<td>Product of efficiency factors of the well and all its superior groups</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WALQ</td>
<td>Well Artificial Lift Quantity</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WMVFP</td>
<td>VFP table number used by the well</td>
</tr>
<tr>
<td>x</td>
<td>FNLPR</td>
<td>GNLPR</td>
<td>WNLPR</td>
<td></td>
<td>FNLPR</td>
<td>NGL Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>FNLPT</td>
<td>GNLPT</td>
<td>WNLPT</td>
<td></td>
<td>FNLPT</td>
<td>NGL Production Total</td>
</tr>
<tr>
<td>x</td>
<td>FNLPRH</td>
<td>GNLPRH</td>
<td>WNLPRH</td>
<td></td>
<td>FNLPRH</td>
<td>NGL Production Rate History (WCONHIST)</td>
</tr>
<tr>
<td>x</td>
<td>FNLPTH</td>
<td>GNLPTH</td>
<td>WNLPTH</td>
<td></td>
<td>FNLPTH</td>
<td>NGL Production Total History (WCONHIST)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>WNLPRRT</td>
<td>NGL Production Rate Target</td>
</tr>
<tr>
<td>E100</td>
<td>E300</td>
<td>Field</td>
<td>Group</td>
<td>Well</td>
<td>Connection</td>
<td>Information</td>
</tr>
<tr>
<td>------</td>
<td>------</td>
<td>-------</td>
<td>-------</td>
<td>------</td>
<td>------------</td>
<td>-------------</td>
</tr>
<tr>
<td>x</td>
<td>FAMF</td>
<td>GAMF</td>
<td>WAMF</td>
<td>CAMF</td>
<td></td>
<td>Component aqueous mole fraction, from producing completions. Note that the mnemonic CAMF is only compatible with the multi-component water COMPW option. The other mnemonics are compatible with the CO2SOL / H2SSOL / GASSOL / KWTABTn / CO2STORE with THERMAL options, in addition to the multi-component water option. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FXMF</td>
<td>GXMF</td>
<td>WXMF</td>
<td></td>
<td></td>
<td>Liquid Mole Fraction. Group and Field level mnemonics only include production wells. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FYMF</td>
<td>GYMF</td>
<td>WYMF</td>
<td></td>
<td></td>
<td>Vapor Mole Fraction. Group and Field level mnemonics only include production wells. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FXMFn</td>
<td>GXMFn</td>
<td>WXMFn</td>
<td></td>
<td></td>
<td>Liquid Mole Fraction for nth separator stage (n = 1) to 99. Group and Field level mnemonics only include production wells. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FYMFn</td>
<td>GYMFn</td>
<td>WYMFn</td>
<td></td>
<td></td>
<td>Vapor Mole Fraction for nth separator stage (n = 1) to 99. Group and Field level mnemonics only include production wells. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FZMF</td>
<td>GZMF</td>
<td>WZMF</td>
<td>CZMF</td>
<td></td>
<td>Total Mole Fraction. Group and Field level mnemonics only include production wells. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCMPR</td>
<td>GCMPR</td>
<td>WCMPR</td>
<td></td>
<td></td>
<td>Hydrocarbon Component Molar Production Rates. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCMPT</td>
<td>GCMPT</td>
<td>WCMPT</td>
<td></td>
<td></td>
<td>Hydrocarbon Component Molar Production Totals. For a well that is both injecting and producing, the total component molar production has to exceed the total component molar injection before a value is reported. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCMIR</td>
<td>GCMIR</td>
<td>WCMIR</td>
<td></td>
<td></td>
<td>Hydrocarbon Component Molar Injection Rates. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCMIT</td>
<td>GCMIT</td>
<td>WCMIT</td>
<td></td>
<td></td>
<td>Hydrocarbon Component Molar Injection Totals. For a well that is both producing and injecting, the total component molar injection has to exceed the total component molar production for a value to be reported. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WCGIR</td>
<td></td>
<td></td>
<td>Hydrocarbon Component Gas Injection Rate. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WCGPR</td>
<td></td>
<td></td>
<td>Hydrocarbon Component Gas Production Rate. See Note 1.</td>
</tr>
<tr>
<td>E100</td>
<td>E300</td>
<td>Field</td>
<td>Group</td>
<td>Well</td>
<td>Connection</td>
<td>Information</td>
</tr>
<tr>
<td>------</td>
<td>------</td>
<td>-------</td>
<td>-------</td>
<td>------</td>
<td>------------</td>
<td>-------------</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>W COPR</td>
<td></td>
<td>Hydrocarbon Component Oil Production Rate. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FHMIR</td>
<td>GHMIR</td>
<td>WHMIR</td>
<td></td>
<td></td>
<td>Hydrocarbon Molar Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>FHMIT</td>
<td>GHMIT</td>
<td>WHMIT</td>
<td></td>
<td></td>
<td>Hydrocarbon Molar Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>FHMPR</td>
<td>GHMPR</td>
<td>WHMPR</td>
<td></td>
<td></td>
<td>Hydrocarbon Molar Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>FHMPT</td>
<td>GHMPT</td>
<td>WHMPT</td>
<td></td>
<td></td>
<td>Hydrocarbon Molar Production Total</td>
</tr>
<tr>
<td>x</td>
<td>FCHMR</td>
<td>GCHMR</td>
<td>WCHMR</td>
<td>CKFR</td>
<td></td>
<td>Hydrocarbon Component Molar Flow Rate. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCHMT</td>
<td>GCHMT</td>
<td>WCHMT</td>
<td>CKFT</td>
<td></td>
<td>Hydrocarbon Component Molar Flow Total. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCWGPR</td>
<td>GCWGPR</td>
<td>WCWGPR</td>
<td></td>
<td></td>
<td>Hydrocarbon Component Wet Gas Production Rate. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCWGPT</td>
<td>GCWGPT</td>
<td>WCWGPT</td>
<td></td>
<td></td>
<td>Hydrocarbon Component Wet Gas Production Total. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCWGIR</td>
<td>GCWGIR</td>
<td>WCWGIR</td>
<td></td>
<td></td>
<td>Hydrocarbon Component Wet Gas Injection Rate. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCWGIT</td>
<td>GCWGIT</td>
<td>WCWGIT</td>
<td></td>
<td></td>
<td>Hydrocarbon Component Wet Gas Injection Total. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCGMR</td>
<td>GCGMR</td>
<td>WCGMR</td>
<td></td>
<td></td>
<td>Hydrocarbon component molar rates in the gas phase. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCGMT</td>
<td>GCGMT</td>
<td>WCGMT</td>
<td></td>
<td></td>
<td>Hydrocarbon component molar totals in the gas phase. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCOMR</td>
<td>GCOMR</td>
<td>WCOMR</td>
<td></td>
<td></td>
<td>Hydrocarbon component molar rates in the oil phase. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCOMT</td>
<td>GCOMT</td>
<td>WCOMT</td>
<td></td>
<td></td>
<td>Hydrocarbon component molar totals in the oil phase. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCNMR</td>
<td>GCNMR</td>
<td>WCNMR</td>
<td></td>
<td></td>
<td>Hydrocarbon component molar rates in the NGL phase. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCNWR</td>
<td>GCNWR</td>
<td>WCNWR</td>
<td></td>
<td></td>
<td>Hydrocarbon component mass rates in the NGL phase</td>
</tr>
<tr>
<td>x</td>
<td>FCGMRn</td>
<td>GCGMRn</td>
<td>WCGMRn</td>
<td></td>
<td></td>
<td>Hydrocarbon component molar rates in the gas phase for nth separator stage (n = 1 to 9). See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCGRn</td>
<td>GCGRn</td>
<td>WCGRn</td>
<td></td>
<td></td>
<td>Hydrocarbon component molar rates in the gas phase for nth separator stage (n = 1 to 99). See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCOMRn</td>
<td>GCOMRn</td>
<td>WCOMRn</td>
<td></td>
<td></td>
<td>Hydrocarbon component molar rates in the oil phase for nth separator stage (n = 1 to 9). See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FCORn</td>
<td>GCORn</td>
<td>WCORn</td>
<td></td>
<td></td>
<td>Hydrocarbon component molar rates in the oil phase for nth separator stage (n = 1 to 99). See Note 1.</td>
</tr>
</tbody>
</table>
### Make-up fraction (calculated at reinjecting wells)

- **E100**: FMUF
- **E300**: GMUF
- **Field**: WMUF

**Information**: Make-up fraction (calculated at reinjecting wells)

### Make-up gas rate (calculated at source of reinjection gas)

- **E100**: FAMR
- **E300**: GAMR

**Information**: Make-up gas rate (calculated at source of reinjection gas)

### Make-up gas total (calculated at source of reinjection gas)

- **E100**: FAMT
- **E300**: GAMT

**Information**: Make-up gas total (calculated at source of reinjection gas)

### D-factor for flow dependent skin factor

- **Field**: CDFAC

**Information**: D-factor for flow dependent skin factor

### Tubing Head Temperature

- **Field**: WTHT

**Information**: Tubing Head Temperature

### Mean molecular weight of wellstream

- **Field**: WMMW

**Information**: Mean molecular weight of wellstream

### Target sustainable rate for most recent sustainable capacity test for gas

- **Field**: FGSPR
- **Group**: GGSPR

**Information**: Target sustainable rate for most recent sustainable capacity test for gas.

### Maximum tested rate sustained for the test period during the most recent sustainable capacity test for gas

- **Field**: FGSRL
- **Group**: GGSRL

**Information**: Maximum tested rate sustained for the test period during the most recent sustainable capacity test for gas.

### Minimum tested rate not sustained for the test period during the most recent sustainable capacity test for gas

- **Field**: FGSRU
- **Group**: GGSRU

**Information**: Minimum tested rate not sustained for the test period during the most recent sustainable capacity test for gas.

### Period for which target sustainable rate could be maintained for the most recent sustainable capacity test for gas. If the target sustainable rate was maintained for the entire test period, this is equal to the test period.

- **Field**: FGSSP
- **Group**: GGSSP

**Information**: Period for which target sustainable rate could be maintained for the most recent sustainable capacity test for gas. If the target sustainable rate was maintained for the entire test period, this is equal to the test period.

### Test period for the most recent sustainable capacity test for gas

- **Field**: FGSTP
- **Group**: GGSTP

**Information**: Test period for the most recent sustainable capacity test for gas.

### Target sustainable rate for most recent sustainable capacity test for oil

- **Field**: FOSPR
- **Group**: GOSPR

**Information**: Target sustainable rate for most recent sustainable capacity test for oil.

### Maximum tested rate sustained for the test period during the most recent sustainable capacity test for oil

- **Field**: FOSRL
- **Group**: GOSRL

**Information**: Maximum tested rate sustained for the test period during the most recent sustainable capacity test for oil.

### Minimum tested rate not sustained for the test period during the most recent sustainable capacity test for oil

- **Field**: FOSRU
- **Group**: GOSRU

**Information**: Minimum tested rate not sustained for the test period during the most recent sustainable capacity test for oil.

### Period for which target sustainable rate could be maintained for the most recent sustainable capacity test for oil. If the target sustainable rate was maintained for the entire test period, this is equal to the test period.

- **Field**: FOSSP
- **Group**: GOSSP

**Information**: Period for which target sustainable rate could be maintained for the most recent sustainable capacity test for oil. If the target sustainable rate was maintained for the entire test period, this is equal to the test period.

### Test period for the most recent sustainable capacity test for oil

- **Field**: FOSTP
- **Group**: GOSTP

**Information**: Test period for the most recent sustainable capacity test for oil.

### Target sustainable rate for most recent sustainable capacity test for water

- **Field**: FWSPR
- **Group**: GWSPR

**Information**: Target sustainable rate for most recent sustainable capacity test for water.
<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>FWSRL</td>
<td>GWSRL</td>
<td></td>
<td></td>
<td>Maximum tested rate sustained for the test period during the most recent sustainable capacity test for water.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FWSRU</td>
<td>GWSRU</td>
<td></td>
<td></td>
<td>Minimum tested rate not sustained for the test period during the most recent sustainable capacity test for water.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FWSSP</td>
<td>GWSSP</td>
<td></td>
<td></td>
<td>Period for which target sustainable rate could be maintained for the most recent sustainable capacity test for water. If the target sustainable rate was maintained for the entire test period, this is equal to the test period.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FWSTP</td>
<td>GWSTP</td>
<td></td>
<td></td>
<td>Test period for the most recent sustainable capacity test for water.</td>
</tr>
<tr>
<td></td>
<td>x</td>
<td>FGPRG</td>
<td>GGPRG</td>
<td></td>
<td></td>
<td>Gas production rate (production summed then separated)</td>
</tr>
<tr>
<td></td>
<td>x</td>
<td>FOPRG</td>
<td>GOPRG</td>
<td></td>
<td></td>
<td>Oil production rate (production summed then separated)</td>
</tr>
<tr>
<td></td>
<td>x</td>
<td>FNLP</td>
<td>GNLP</td>
<td></td>
<td></td>
<td>NGL production rate (production summed then separated)</td>
</tr>
<tr>
<td></td>
<td>x</td>
<td>FXMFG</td>
<td>GXMFG</td>
<td></td>
<td></td>
<td>Liquid mole fraction (production summed then separated). See Note 1.</td>
</tr>
<tr>
<td></td>
<td>x</td>
<td>FYMFG</td>
<td>GYMFG</td>
<td></td>
<td></td>
<td>Vapor mole fraction (production summed then separated). See Note 1.</td>
</tr>
<tr>
<td></td>
<td>x</td>
<td>FCOMRG</td>
<td>GCOMRG</td>
<td></td>
<td></td>
<td>Hydrocarbon component molar rates in the oil phase (production summed then separated). See Note 1.</td>
</tr>
<tr>
<td></td>
<td>x</td>
<td>FCGMRG</td>
<td>GCGMRG</td>
<td></td>
<td></td>
<td>Hydrocarbon component molar rates in the gas phase (production summed then separated). See Note 1.</td>
</tr>
<tr>
<td></td>
<td>x</td>
<td>FCNMRG</td>
<td>GCNMRG</td>
<td></td>
<td></td>
<td>Hydrocarbon component molar rates in the NGL phase (production summed then separated). See Note 1.</td>
</tr>
<tr>
<td></td>
<td>x</td>
<td></td>
<td>WPWE0</td>
<td></td>
<td></td>
<td>Well drilled indicator. Set to one if the well was drilled this timestep and zero at other times.</td>
</tr>
<tr>
<td></td>
<td>x</td>
<td></td>
<td>WPWE1</td>
<td></td>
<td></td>
<td>Connections opened indicator. Set to the number of connections opened in the well this timestep if the well is not shut or stopped.</td>
</tr>
<tr>
<td></td>
<td>x</td>
<td></td>
<td>WPWE2</td>
<td></td>
<td></td>
<td>Connections closed indicator. Set to the number of connections closed in the well this timestep. This will not include connections closed by a CON+ workover.</td>
</tr>
</tbody>
</table>
### Table 2.25: Miscellaneous well and group quantities

**ECLIPSE 300 Thermal**

The miscellaneous well quantity options are:

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WPWE3</td>
<td>Connections closed to bottom indicator. Set to 1 if all connections are closed by CON workovers, or if all connections below a connection are closed by a CON+ workover. Set to zero at other timesteps.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WPWE4</td>
<td>Well stopped indicator. Set to one if the well is stopped this timestep and zero otherwise.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WPWE5</td>
<td>Injector to producer indicator. Set to one if the well is switched from being an injector to a producer this timestep. Set to zero otherwise.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WPWE6</td>
<td>Producer to injector indicator. Set to one if the well is switched from being a producer to an injector this timestep. Set to zero otherwise.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WPWE7</td>
<td>Well shut indicator. Set to one if the well is shut this timestep and zero otherwise.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WPWEM</td>
<td>WELEVNT output mnemonic.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WDRPR</td>
<td>Well drilling priority.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WBHWCn</td>
<td>Derivative of well BHP with respect to parameter n (n=1 to 99). See GWRTWCV.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WGFWCn</td>
<td>Derivative of well gas flow rate with respect to parameter n, n=1 to 99. See GWRTWCV.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WOFWCn</td>
<td>Derivative of well oil flow rate with respect to parameter n, n=1 to 99. See GWRTWCV.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>WWFWCn</td>
<td>Derivative of water flow rate with respect to parameter n, n=1 to 99. See GWRTWCV.</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WSQU</td>
<td>Injected steam quality</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WTEMP</td>
<td>Temperature</td>
</tr>
<tr>
<td>x</td>
<td>FSTPR</td>
<td>GSTPR</td>
<td>WSTPR</td>
<td>Steam production rate</td>
</tr>
<tr>
<td>x</td>
<td>FSTPT</td>
<td>GSTPT</td>
<td>WSTPT</td>
<td>Steam production total</td>
</tr>
<tr>
<td>x</td>
<td>FOSRC</td>
<td>GOSRC</td>
<td>Cumulative oil steam ratio</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FSORC</td>
<td>GSORC</td>
<td>Cumulative steam oil ratio</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FHSRC</td>
<td>GHSRC</td>
<td>Cumulative hydrocarbon steam ratio</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FSHRC</td>
<td>GSHRC</td>
<td>Cumulative steam hydrocarbon ratio</td>
<td></td>
</tr>
</tbody>
</table>

_Table 2.26: Miscellaneous well quantities (ECLIPSE 300 Thermal)_
## Secondary group and well quantities (ECLIPSE 300)

The secondary group and well quantities are:

<table>
<thead>
<tr>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDOPR</td>
<td>Secondary well oil production rate.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDWPR</td>
<td>Secondary well water production rate.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDGPR</td>
<td>Secondary well gas production rate.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDLPR</td>
<td>Secondary well liquid production rate.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDVPR</td>
<td>Secondary well reservoir volume production rate.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDOPT</td>
<td>Secondary well oil production total.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDWPT</td>
<td>Secondary well water production total.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDGPT</td>
<td>Secondary well gas production total.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDLPT</td>
<td>Secondary well liquid production total.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDVPT</td>
<td>Secondary well reservoir volume production total.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDWIR</td>
<td>Secondary well water injection rate.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDGIR</td>
<td>Secondary well gas injection rate.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDWIT</td>
<td>Secondary well water injection total.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDGIT</td>
<td>Secondary well gas injection total.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDOGR</td>
<td>Secondary well oil-gas ratio.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDGOR</td>
<td>Secondary well gas-oil ratio.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDGLR</td>
<td>Secondary well gas-liquid ratio.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDWGR</td>
<td>Secondary well water-gas ratio.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDWCT</td>
<td>Secondary well water cut.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDTEM</td>
<td>Secondary well temperature.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>WDPR</td>
<td>Secondary well pressure.</td>
</tr>
<tr>
<td>x</td>
<td>FDOPR</td>
<td>GDOPR</td>
<td>Oil production rate including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDWPR</td>
<td>GDWPR</td>
<td>Water production rate including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDGPR</td>
<td>GDGPR</td>
<td>Gas production rate including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDLPR</td>
<td>GDLPR</td>
<td>Liquid production rate including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDVPR</td>
<td>GDVPR</td>
<td>Reservoir volume production rate including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDOPT</td>
<td>GDOPT</td>
<td>Oil production total including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDWPT</td>
<td>GDWPT</td>
<td>Water production total including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDGPT</td>
<td>GDGPT</td>
<td>Gas production total including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDLPT</td>
<td>GDLPT</td>
<td>Liquid production total including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDWIR</td>
<td>GDWIR</td>
<td>Water injection rate including secondary well contributions</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDGIR</td>
<td>GDGIR</td>
<td>Gas injection rate including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDWIT</td>
<td>GDWIT</td>
<td>Water injection total including secondary well contributions.</td>
<td></td>
</tr>
</tbody>
</table>
### Table 2.27: Secondary group and well quantities (ECLIPSE 300)

<table>
<thead>
<tr>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>FDGIT</td>
<td>GDGIT</td>
<td>Gas injection total including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDGGR</td>
<td>GDGGR</td>
<td>Oil-gas ratio including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDOGR</td>
<td>GDGGR</td>
<td>Gas-oil ratio including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDGLR</td>
<td>GDGLR</td>
<td>Gas-liquid ratio including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDWGR</td>
<td>GDWGR</td>
<td>Water-gas ratio including secondary well contributions.</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FDWCT</td>
<td>GDWCT</td>
<td>Water cut including secondary well contributions.</td>
<td></td>
</tr>
</tbody>
</table>

### Well list quantities

The well list quantity options are:

<table>
<thead>
<tr>
<th>E300</th>
<th>Well list</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>GOPR</td>
<td>Oil Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>GOPT</td>
<td>Oil Production Total</td>
</tr>
<tr>
<td>x</td>
<td>GWPR</td>
<td>Water Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>GWPT</td>
<td>Water Production Total</td>
</tr>
<tr>
<td>x</td>
<td>GWIR</td>
<td>Water Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>GWIT</td>
<td>Water Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>GGPR</td>
<td>Gas Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>GGPT</td>
<td>Gas Production Total</td>
</tr>
<tr>
<td>x</td>
<td>GGIR</td>
<td>Gas Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>GGIT</td>
<td>Gas Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>GVPR</td>
<td>Voidage Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>GVPT</td>
<td>Voidage Production Total</td>
</tr>
<tr>
<td>x</td>
<td>GVIR</td>
<td>Voidage Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>GVIT</td>
<td>Voidage Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>GLPR</td>
<td>Liquid Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>GLPT</td>
<td>Liquid Production Total</td>
</tr>
<tr>
<td>x</td>
<td>GGOR</td>
<td>Gas to Oil Ratio</td>
</tr>
<tr>
<td>x</td>
<td>GOGR</td>
<td>Oil to Gas Ratio</td>
</tr>
<tr>
<td>x</td>
<td>GWCT</td>
<td>Water Cut</td>
</tr>
<tr>
<td>x</td>
<td>GWGR</td>
<td>Water to Gas Ratio</td>
</tr>
<tr>
<td>x</td>
<td>GGLR</td>
<td>Gas to Liquid Ratio</td>
</tr>
<tr>
<td>x</td>
<td>GOPRH</td>
<td>Oil Production Rate History (WCONHIST)</td>
</tr>
<tr>
<td>x</td>
<td>GGPRH</td>
<td>Gas Production Rate History (WCONHIST)</td>
</tr>
<tr>
<td>x</td>
<td>GWPRH</td>
<td>Water Production Rate History (WCONHIST)</td>
</tr>
<tr>
<td>x</td>
<td>GLPRH</td>
<td>Liquid Production Rate History (WCONHIST)</td>
</tr>
</tbody>
</table>
Table 2.28: Well list quantities (ECLIPSE 300)

**Grid block and region quantities**

The grid block and region oil quantity options given in the table below. Notes for this table:

1. See "Initial fluid in place".
2. See “Format of SUMMARY keywords with a component index”.

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>BOSAT</td>
<td></td>
<td></td>
<td>Oil SATuration</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>BSOIL</td>
<td></td>
<td></td>
<td>Oil Saturation (same as BOSAT)</td>
</tr>
<tr>
<td>x</td>
<td>BSOILM</td>
<td></td>
<td></td>
<td></td>
<td>Mobile Oil Saturation (SOR option)</td>
</tr>
<tr>
<td>x</td>
<td>BSOILR</td>
<td></td>
<td></td>
<td></td>
<td>Residual Oil Saturation (SOR option)</td>
</tr>
<tr>
<td>x</td>
<td>BCOILR</td>
<td></td>
<td></td>
<td></td>
<td>Residual Oil compressibility (SOR option)</td>
</tr>
<tr>
<td>x</td>
<td>FOIP</td>
<td>ROIP</td>
<td>BOIP</td>
<td></td>
<td>Oil In Place (in liquid and wet gas phases). See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FOIPL</td>
<td>ROIPL</td>
<td>BOIPL</td>
<td></td>
<td>Oil In Place (Liquid phase). See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FOIPG</td>
<td>ROIPG</td>
<td>BOIPG</td>
<td></td>
<td>Oil In Place (Gas phase). See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FOIPR</td>
<td></td>
<td></td>
<td></td>
<td>Oil Reservoir Volume in Place</td>
</tr>
<tr>
<td>x</td>
<td>FCOM</td>
<td>RCOM</td>
<td></td>
<td></td>
<td>Molar amount of specified component dissolved in oil. See Note 2.</td>
</tr>
<tr>
<td>x</td>
<td>FOIPA</td>
<td>ROIPA</td>
<td></td>
<td></td>
<td>Oil in Place above TRACK depth. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FOIPB</td>
<td>ROIPB</td>
<td></td>
<td></td>
<td>Oil in Place below TRACK depth. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>FPPO</td>
<td>RPPO</td>
<td>BPPO</td>
<td></td>
<td>Oil Potential (or “depth corrected pressure”)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>BVOIL</td>
<td></td>
<td></td>
<td>Oil Viscosity</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>BOVIS</td>
<td></td>
<td></td>
<td>Oil VIScosity (same as BVOIL)</td>
</tr>
<tr>
<td>x</td>
<td>FOVIS</td>
<td>ROVIS</td>
<td></td>
<td></td>
<td>Oil VIScosity average value</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>BDENO</td>
<td></td>
<td></td>
<td>Oil reservoir Density</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>BODEN</td>
<td></td>
<td></td>
<td>Oil reservoir DENSity (same as BDENO)</td>
</tr>
</tbody>
</table>
### Table 2.29: Grid block and region oil quantities

The grid block and region water quantity options are given in the table below. Notes for this table:

1. See “Format of SUMMARY keywords with a component index”.

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td>FODEN</td>
<td>RODEN</td>
<td></td>
<td>Oil reservoir DENsity average value</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>BBOIL</td>
<td></td>
<td></td>
<td>Oil Reservoir Molar Densities</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>BFUGO</td>
<td></td>
<td></td>
<td>Hydrocarbon component FUGacities in Oil (DIFFUSE option only, see Note 2).</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>ROP</td>
<td></td>
<td></td>
<td>Net oil production from a region</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>ROPR</td>
<td></td>
<td></td>
<td>Oil Production Rate. In ECLIPSE 300, this is calculated as the sum of (connection molar rate multiplied by well separator oil rate per mole) for all connections in the region.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>ROPT</td>
<td></td>
<td></td>
<td>Oil Production Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>ROIR</td>
<td></td>
<td></td>
<td>Oil Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>ROI</td>
<td></td>
<td></td>
<td>Oil Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>BFLOOI</td>
<td></td>
<td></td>
<td>Inter-block oil flow rate in the positive I direction. This value is, by default, calculated based on the first FIP set.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>BFLOOJ</td>
<td></td>
<td></td>
<td>Inter-block oil flow rate in the positive J direction. This value is, by default, calculated based on the first FIP set.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>BFLOOK</td>
<td></td>
<td></td>
<td>Inter-block oil flow rate in the positive K direction. This value is, by default, calculated based on the first FIP set.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>BVELOI</td>
<td></td>
<td></td>
<td>Oil velocity for positive I direction.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>BVELOJ</td>
<td></td>
<td></td>
<td>Oil velocity for positive J direction.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>BVELOK</td>
<td></td>
<td></td>
<td>Oil velocity for positive K direction.</td>
</tr>
</tbody>
</table>
Table 2.30: Grid block and region water quantities

The grid block and region gas quantity options are given in the table below. Notes for this table:

1. See "Initial fluid in place”.
2. See “Format of SUMMARY keywords with a component index”.

### Table 2.30: Grid block and region water quantities

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td>FWVIS</td>
<td>RWVIS</td>
<td>x</td>
<td>Water VIScosity average value</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>BDENW</td>
<td></td>
<td>x</td>
<td>Water reservoir Density</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>BWDEN</td>
<td></td>
<td>x</td>
<td>Water reservoir DENsity (same as BDENW)</td>
</tr>
<tr>
<td>x</td>
<td>FWDEN</td>
<td>RWDEN</td>
<td></td>
<td>x</td>
<td>Water reservoir DENsity average value</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>BBWAT</td>
<td></td>
<td>x</td>
<td>Water Reservoir Molar Densities</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>BXXWAT</td>
<td></td>
<td>x</td>
<td>Fractional Saturation of Water (Gravity Drainage Option only)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RWP</td>
<td></td>
<td>x</td>
<td>Net water production from a region (minus injection from wells and analytic aquifers)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>RWPR</td>
<td></td>
<td>x</td>
<td>Water Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>RWPT</td>
<td></td>
<td>x</td>
<td>Water Production Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>RWIR</td>
<td></td>
<td>x</td>
<td>Water Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>RWIT</td>
<td></td>
<td>x</td>
<td>Water Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>BFLOWI</td>
<td></td>
<td>x</td>
<td>Inter-block water flow rate in the positive I direction.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>BFLOWJ</td>
<td></td>
<td>x</td>
<td>Inter-block water flow rate in the positive J direction.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>BFLOWK</td>
<td></td>
<td>x</td>
<td>Inter-block water flow rate in the positive K direction.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>BVELWI</td>
<td></td>
<td>x</td>
<td>Water velocity in the positive I direction.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>BVELWJ</td>
<td></td>
<td>x</td>
<td>Water velocity in the positive J direction.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>BVELWK</td>
<td></td>
<td>x</td>
<td>Water velocity in the positive K direction.</td>
</tr>
<tr>
<td>Field</td>
<td>Region</td>
<td>Block</td>
<td>Information</td>
<td></td>
<td></td>
</tr>
<tr>
<td>--------</td>
<td>--------</td>
<td>-------</td>
<td>-------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FCGMI</td>
<td>RCGMI</td>
<td></td>
<td>Molar amount of specified component trapped in gas. See Note 2.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FCGMM</td>
<td>RCGMM</td>
<td></td>
<td>Molar amount of specified component mobile in gas. See Note 2.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FGCDI</td>
<td>RGCDI</td>
<td></td>
<td>CO₂ (“carbon dioxide”) trapped (immobile) in gas phase (CO2STORE option only)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FGCDM</td>
<td>RGCDM</td>
<td></td>
<td>CO₂ (“carbon dioxide”) mobile in gas phase (CO2STORE option only)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FPPG</td>
<td>RPPG</td>
<td>BPPG</td>
<td>Gas Potential (or “depth corrected pressure”)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BVGAS</td>
<td></td>
<td></td>
<td>Gas Viscosity</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FGVIS</td>
<td>RGVIS</td>
<td></td>
<td>Gas Viscosity average value</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BDENG</td>
<td></td>
<td></td>
<td>Gas reservoir Density</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BGDEN</td>
<td></td>
<td></td>
<td>Gas reservoir Density (same as BDENG)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FGDEN</td>
<td>RGDEN</td>
<td></td>
<td>Gas reservoir Density average value</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BBGAS</td>
<td></td>
<td></td>
<td>Gas Reservoir Molar Densities</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BFUGG</td>
<td></td>
<td></td>
<td>Hydrocarbon component FUGacities in Gas (DIFFUSE option only, see Note 2)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BXGAS</td>
<td></td>
<td></td>
<td>Fractional Saturation of Gas (Gravity Drainage Option only)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RGP</td>
<td></td>
<td></td>
<td>Net gas production from a region (minus injection)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RGPR</td>
<td></td>
<td></td>
<td>Gas Production Rate. In ECLIPSE 300, this is calculated as the sum of (connection molar rate multiplied by well separator gas rate per mole) for all connections in the region.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RGPFRF</td>
<td></td>
<td></td>
<td>Free Gas Production Rate</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RGPRS</td>
<td></td>
<td></td>
<td>Solution Gas Production Rate</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RGPT</td>
<td></td>
<td></td>
<td>Gas Production Total</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RGPTF</td>
<td></td>
<td></td>
<td>Free Gas Production Total</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RGPTS</td>
<td></td>
<td></td>
<td>Solution Gas Production Total</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RGIR</td>
<td></td>
<td></td>
<td>Gas Injection Rate</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RGIT</td>
<td></td>
<td></td>
<td>Gas Injection Total</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RNLPR</td>
<td></td>
<td></td>
<td>NGL Production Rate</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RNLPT</td>
<td></td>
<td></td>
<td>NGL Production Total</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BFLOGI</td>
<td></td>
<td></td>
<td>Inter-block gas flow rate in the positive I direction. This value is, by default, calculated based on the first FIP set.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BFLOGJ</td>
<td></td>
<td></td>
<td>Inter-block gas flow rate in the positive J direction. This value is, by default, calculated based on the first FIP set.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BFLOGK</td>
<td></td>
<td></td>
<td>Inter-block gas flow rate in the positive K direction. This value is, by default, calculated based on the first FIP set.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BVELGI</td>
<td></td>
<td></td>
<td>Gas velocity in the positive I direction.</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Table 2.31: Grid block and region gas quantities

The grid block solid quantity options are:

<table>
<thead>
<tr>
<th>E300</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>BBSOL</td>
<td>Solid reservoir molar density</td>
</tr>
<tr>
<td>x</td>
<td>BDENS</td>
<td>Solid Density</td>
</tr>
<tr>
<td>x</td>
<td>BHSOL</td>
<td>Solid molar enthalpy (Thermal option)</td>
</tr>
<tr>
<td>x</td>
<td>BSDEN</td>
<td>Solid DENsity (same as BDENS)</td>
</tr>
<tr>
<td>x</td>
<td>BSOLADS</td>
<td>Adsorbed solid concentration</td>
</tr>
<tr>
<td>x</td>
<td>BSOLADW</td>
<td>Adsorbed wetting-solid concentration</td>
</tr>
<tr>
<td>x</td>
<td>BSOLWET</td>
<td>Solid wettability interpolation parameter</td>
</tr>
<tr>
<td>x</td>
<td>BSSAT</td>
<td>Solid SATuration</td>
</tr>
<tr>
<td>x</td>
<td>BSSOLID</td>
<td>Solid Saturation (same as BSSAT)</td>
</tr>
<tr>
<td>x</td>
<td>BSSFrac</td>
<td>Suspended Solid FRACtion</td>
</tr>
<tr>
<td>x</td>
<td>BSMMULT</td>
<td>Solid Mobility MULTIplier</td>
</tr>
</tbody>
</table>

Table 2.32: Grid block solid quantities (ECLIPSE 300)

The block mobility quantity options are: given in the table below. Notes for this table:
1. See “Format of SUMMARY keywords with a component index”.

<table>
<thead>
<tr>
<th>E300</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>BGMCO</td>
<td>Generalized mobilities of hydrocarbon components in oil. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>BGMCG</td>
<td>Generalized mobilities of hydrocarbon components in gas. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>BGMCW</td>
<td>Generalized mobilities of water</td>
</tr>
<tr>
<td>x</td>
<td>BGMRV</td>
<td>Generalized mobilities of reservoir volume</td>
</tr>
</tbody>
</table>

Table 2.33: Block mobility quantities (ECLIPSE 300)

The inter-region flow quantity options are given in the table below. Notes for this table:
1. See “Format of SUMMARY keywords with a component index”.

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Region</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>ROFR</td>
<td>Inter-region oil flow rate</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>ROFR+</td>
<td>Inter-region oil flow rate (+ve contributions)</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>ROFR-</td>
<td>Inter-region oil flow rate (-ve contributions)</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>ROFT</td>
<td>Inter-region oil flow total (in liquid and wet gas phases)</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>ROFT+</td>
<td>Inter-region oil flow total (in liquid and wet gas phases) (+ve contributions)</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>ROFT-</td>
<td>Inter-region oil flow total (in liquid and wet gas phases) (-ve contributions)</td>
<td></td>
</tr>
<tr>
<td>E100</td>
<td>E300</td>
<td>Region</td>
<td>Information</td>
</tr>
<tr>
<td>------</td>
<td>------</td>
<td>----------------</td>
<td>-------------------------------------------------------</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>ROFTL</td>
<td>Inter-region oil flow total (liquid phase)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>ROFTG</td>
<td>Inter-region oil flow total (gas phase)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RGFR</td>
<td>Inter-region gas flow rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RGFR+</td>
<td>Inter-region gas flow rate (+ve contributions)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RGFR-</td>
<td>Inter-region gas flow rate (-ve contributions)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>RGFT</td>
<td>Inter-region gas flow total (in liquid and gas phases)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RGFT+</td>
<td>Inter-region gas flow total (in liquid and gas phases) (+ve contributions)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RGFT-</td>
<td>Inter-region gas flow total (in liquid and gas phases) (-ve contributions)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>RGFTL</td>
<td>Inter-region gas flow total (liquid phase)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>RGFTG</td>
<td>Inter-region gas flow total (gas phase)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RWFR</td>
<td>Inter-region water flow rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RWFR+</td>
<td>Inter-region water flow rate (+ve contributions)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RWFR-</td>
<td>Inter-region water flow rate (-ve contributions)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>RWFT</td>
<td>Inter-region water flow total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RWFT+</td>
<td>Inter-region water flow total (+ve contributions)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RWFT-</td>
<td>Inter-region water flow total (-ve contributions)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RKFR</td>
<td>Inter-region component flow rates. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RKFR+</td>
<td>Inter-region component flow rates (+ve contributions). See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RKFR-</td>
<td>Inter-region component flow rates (-ve contributions). See Note 1.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>RKFT</td>
<td>Inter-region component flow totals. See Note 1.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>RKFT+</td>
<td>Inter-region component flow totals (+ve contributions). See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RKFT-</td>
<td>Inter-region component flow totals (-ve contributions). See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>REFR</td>
<td>Inter-region energy flow rates (Thermal option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>REFR+</td>
<td>Inter-region energy flow rates (+ve contributions) (Thermal option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>REFR-</td>
<td>Inter-region energy flow rates, (-ve contributions) (Thermal option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>REFT</td>
<td>Inter-region energy flow totals (Thermal option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>REFT+</td>
<td>Inter-region energy flow totals (+ve contributions) (Thermal option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>REFT-</td>
<td>Inter-region energy flow totals (-ve contributions) (Thermal option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RNLF R</td>
<td>Inter-region NGL flow rates</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RNLF R+</td>
<td>Inter-region NGL flow rates (+ve contributions)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RNLF R-</td>
<td>Inter-region NGL flow rates (-ve contributions)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RNLF T</td>
<td>Inter-region NGL flow totals</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RNLF T+</td>
<td>Inter-region NGL flow totals (+ve contributions)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RNLF T-</td>
<td>Inter-region NGL flow totals (-ve contributions)</td>
</tr>
</tbody>
</table>

Table 2.34: Inter-region flow quantities
**Note:** For some rare situations the reported regional flow total and the regional fluid in place changes might show some mismatch. In order to improve the reported value you can use **OPTIONS**3 item 206.

The miscellaneous grid block and region quantity options are given in the table below. Notes for this table:

1. See “Format of SUMMARY keywords with a component index”.
2. See "Aqueous speciation" in the *ECLIPSE Technical Description*

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>BPR</td>
<td>Oil phase Pressure</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BPRESSUR</td>
<td>Oil phase Pressure (same as BPR)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BWPR</td>
<td>Water phase Pressure</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BGR</td>
<td>Gas phase Pressure</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FPR</td>
<td>RPR</td>
<td></td>
<td>Pressure average value (same as FPRH/RPRH unless, for ECLIPSE 100, item 31 of OPTIONS is set)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FPRH</td>
<td>RPRH</td>
<td></td>
<td>Pressure average value (Hydrocarbon Pore-Volume Weighted)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FPRP</td>
<td>RPRP</td>
<td></td>
<td>Pressure average value (Pore-Volume Weighted)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FPRGZ</td>
<td>RPRGZ</td>
<td></td>
<td>P/Z (gas) average value weighted by gas content of blocks at surface conditions</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BRS</td>
<td>Gas-oil ratio (in liquid phase)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FRS</td>
<td>RRS</td>
<td></td>
<td>Gas-oil ratio (in liquid phase) average value</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BRV</td>
<td>Oil-gas ratio (in gas phase)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FRV</td>
<td>RRV</td>
<td></td>
<td>Oil-gas ratio (in gas phase) average value</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FCHIP</td>
<td></td>
<td></td>
<td>Component Hydrocarbon as Wet Gas. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FCMIP</td>
<td></td>
<td></td>
<td>Component Hydrocarbon as Moles. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BPBUB</td>
<td>Bubble point pressure</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BPDEW</td>
<td>Dew point pressure</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BRSSAT</td>
<td>Saturated gas-oil ratio (in oil phase)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BRVSAT</td>
<td>Saturated oil-gas ratio (in gas phase)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BSTATE</td>
<td>Gas-oil state indicator. Hydrocarbon states are: 1 Gas only (with vaporized oil) 2 Gas and Oil 3 Oil only (with dissolved gas)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FPPC</td>
<td>RPPC</td>
<td>BPPC</td>
<td>Initial Contact Corrected Potential</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>BOKR</td>
<td>Oil relative permeability</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>BWKR</td>
<td>Water relative permeability</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>BGKR</td>
<td>Gas relative permeability</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>BKRO</td>
<td>Oil relative permeability</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>BKROG</td>
<td>Two-phase oil relative permeability to gas</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>BKROW</td>
<td>Two-phase oil relative permeability to water</td>
</tr>
<tr>
<td>E100</td>
<td>E300</td>
<td>Field</td>
<td>Region</td>
<td>Block</td>
<td>Information</td>
</tr>
<tr>
<td>------</td>
<td>------</td>
<td>-------</td>
<td>--------</td>
<td>-------</td>
<td>-------------</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>E100</td>
<td></td>
<td>BKRG</td>
<td>Gas relative permeability</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>E300</td>
<td>Field</td>
<td>BKRRGO</td>
<td>Two-phase gas relative permeability to oil (BAKER1, BAKER2 and IKU3P three-phase relative permeability models)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>Region</td>
<td>BKRGW</td>
<td>Two-phase gas relative permeability to water (BAKER1, BAKER2 and IKU3P three-phase relative permeability models)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td>Block</td>
<td>BKRW</td>
<td>Water relative permeability</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BKRWG</td>
<td>Two-phase water relative permeability to gas (BAKER1, BAKER2 and IKU3P three-phase relative permeability models)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BKRW0</td>
<td>Two-phase water relative permeability to oil (BAKER1, BAKER2, and IKU3P three-phase relative permeability models)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BRK</td>
<td>Water relative permeability reduction factor due to polymer</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BEWKR</td>
<td>Water effective relative permeability due to polymer</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>BWFC</td>
<td>Water-Oil capillary pressure</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>BGPC</td>
<td>Gas-Oil capillary pressure</td>
</tr>
<tr>
<td></td>
<td>x</td>
<td></td>
<td></td>
<td>BPCO</td>
<td>Oil Capillary Pressures</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BPCG</td>
<td>Gas Capillary Pressures</td>
</tr>
<tr>
<td></td>
<td>x</td>
<td></td>
<td></td>
<td>BPCW</td>
<td>Water Capillary Pressures</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>BGTRP</td>
<td>Trapped gas saturation (WAG hysteresis only)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>BGTPD</td>
<td>Dynamic trapped gas saturation (WAG hysteresis only)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BGSHY</td>
<td>Departure saturation from drainage to imbibition for gas capillary pressure hysteresis</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BGSTRP</td>
<td>Trapped gas critical saturation for gas capillary pressure hysteresis</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BWSHY</td>
<td>Departure saturation from drainage to imbibition for water capillary pressure hysteresis</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BWSMA</td>
<td>Maximum wetting saturation for water capillary pressure hysteresis. This also represents the maximum wetting phase saturation used to determine scanning curves in relative permeability hysteresis, whether the Carlson or Killough model is used.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BMLSC</td>
<td>Hydrocarbon molar density (moles per reservoir volume). See Note 1. When the multi-component water option is enabled, the water components are allowed as described in the link above.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BMLST</td>
<td>Total hydrocarbon molar density (moles per reservoir volume).</td>
</tr>
<tr>
<td>E100</td>
<td>E300</td>
<td>Field</td>
<td>Region</td>
<td>Block</td>
<td>Information</td>
</tr>
<tr>
<td>------</td>
<td>------</td>
<td>-------</td>
<td>--------</td>
<td>--------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BMWAT</td>
<td>Water molar density (moles per reservoir volume). When the multi-component water option is enabled, this relates to the last component.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BROMLS</td>
<td>Residual oil moles/reservoir volume (SOR option). See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BJV</td>
<td>In (K) values. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BVMF</td>
<td>Vapor mole fraction.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BPSAT</td>
<td>Saturation Pressures. The value depends on the cell hydrocarbon state: for single phase gas it will usually be the dew point pressure; for single phase oil it will usually be the bubble point pressure; for two phase cells it will be the cell pressure.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BAMF</td>
<td>Component aqueous mole fraction. (KWTABTn, or CO2STORE with THERMAL, or CO2SOL/H2SSOL/GASSOL and with multi-component water COMPW options). See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BXMF</td>
<td>Liquid hydrocarbon component mole fraction. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BYMF</td>
<td>Vapor hydrocarbon component mole fraction / vapor steam (THERMAL option) mole fraction. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BSMF</td>
<td>(CO2STORE with SOLID option only) Solid hydrocarbon component mole fraction. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BSTEN</td>
<td>Surface Tension.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BFMISC</td>
<td>Miscibility Factor.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>FREAC</td>
<td>BREAC</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>FREAT</td>
<td>Reaction total. The reaction number is given as a component index. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BHD</td>
<td>Hydraulic head.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BHDF</td>
<td>Hydraulic head at fresh water conditions.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BPR_X</td>
<td>Pressure interpolated at a defined coordinate.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BHD_X</td>
<td>Hydraulic head interpolated at a defined coordinate.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BHDF_X</td>
<td>Hydraulic head at fresh water conditions interpolated at a defined coordinate.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BSCN_X</td>
<td>Brine concentration interpolated at a defined coordinate.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BCTRA_X</td>
<td>Tracer concentration interpolated at a defined coordinate. Implies a tracer name TRA and this name needs to start with T</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>LBPR_X</td>
<td>Pressure interpolated at a defined coordinate within a local grid</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>LBHD_X</td>
<td>Hydraulic head interpolated at a defined coordinate within a local grid</td>
</tr>
</tbody>
</table>
### Table 2.35: Miscellaneous grid block and region quantities

The field, grid block and region quantity options are:

<table>
<thead>
<tr>
<th>E300</th>
<th>Field</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>LBHDF_X</td>
<td>Hydraulic head at freshwater conditions interpolated at a defined coordinate within a local grid.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>LBSCN_X</td>
<td>Brine concentration interpolated at a defined coordinate within a local grid.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>LBCTRA_X</td>
<td>Tracer concentration interpolated at a defined coordinate within a local grid. Implies a tracer name TRA and this name needs to start with T</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BAQSP</td>
<td>((\text{CO2STORE} \text{ option only})) Molar density of ions. See Note 2. The species number is given as component index. See Note 1. If no component number is given, all species are selected.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BAQPH</td>
<td>((\text{CO2STORE} \text{ option only})) Aqueous pH level. See Note 2.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BBTFORG</td>
<td>Forchheimer beta factor for gas phase.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BBTFORO</td>
<td>Forchheimer beta factor for oil phase.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BKFRI</td>
<td>Inter-block component flow rate in the positive I direction. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BKFRJ</td>
<td>Inter-block component flow rate in the positive J direction. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BKFRK</td>
<td>Inter-block component flow rate in the positive K direction. See Note 1.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>E300</th>
<th>Field</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BTEMP</td>
<td>Temperature</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BENERGY</td>
<td>Internal energy/bulk volume</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BHOIL</td>
<td>Molar Enthalpy of Oil</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BHGAS</td>
<td>Molar Enthalpy of Gas</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BHWAT</td>
<td>Molar Enthalpy of Water</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BSPENOIL</td>
<td>Molar Energy of Oil</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BSPENGAS</td>
<td>Molar Energy of Gas</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BSPENWAT</td>
<td>Molar Energy of Water</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BSPENROC</td>
<td>Specific Internal Energy of Rock</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BGMEO</td>
<td>Enthalpy mobility of Oil</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BGMEG</td>
<td>Enthalpy mobility of Gas</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BGMEW</td>
<td>Enthalpy mobility of Water</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BTCMULT</td>
<td>Saturation dependent conductivity factor</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BSFOIL</td>
<td>Saturation of foamy oil. Calculated as the saturation of the components in the oil phase with gas-like compressibility types.</td>
</tr>
</tbody>
</table>
Table 2.36: Field, grid block, region quantities (ECLIPSE 300 Thermal)

The rock quantity options are:

<table>
<thead>
<tr>
<th>E300</th>
<th>Field</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>BSFSOL</td>
<td></td>
<td></td>
<td>Saturation of foamy solids. Calculated as the saturation of the components in the solid phase with gas-like compressibility types.</td>
</tr>
<tr>
<td>x</td>
<td>FERCK</td>
<td>RERCK</td>
<td></td>
<td>Energy in Rock</td>
</tr>
<tr>
<td>x</td>
<td>FEOIL</td>
<td>REOIL</td>
<td></td>
<td>Energy in Oil</td>
</tr>
<tr>
<td>x</td>
<td>FEGAS</td>
<td>REGAS</td>
<td></td>
<td>Energy in Gas</td>
</tr>
<tr>
<td>x</td>
<td>FEWAT</td>
<td>REWAT</td>
<td></td>
<td>Energy in Water</td>
</tr>
<tr>
<td>x</td>
<td>FETOT</td>
<td>RETOT</td>
<td></td>
<td>Total Energy</td>
</tr>
<tr>
<td>x</td>
<td>FEAVE</td>
<td>REAVE</td>
<td></td>
<td>Average Energy Density</td>
</tr>
<tr>
<td>x</td>
<td>FERPC</td>
<td>RERPC</td>
<td></td>
<td>Energy Percentage in Rock</td>
</tr>
<tr>
<td>x</td>
<td>FEOPC</td>
<td>REOPC</td>
<td></td>
<td>Energy Percentage in Oil</td>
</tr>
<tr>
<td>x</td>
<td>FEGPC</td>
<td>REGPC</td>
<td></td>
<td>Energy Percentage in Gas</td>
</tr>
<tr>
<td>x</td>
<td>FEWPC</td>
<td>REWPC</td>
<td></td>
<td>Energy Percentage in Water</td>
</tr>
</tbody>
</table>

Table 2.37: Rock quantities (ECLIPSE 300 Thermal)

The default average field pressure is the hydrocarbon pore volume weighted average defined as:

\[ FPRH = \frac{\sum HCPV \cdot p}{\sum HCPV} \] (Eq. 2.1)

where \( HCPV = PV(1-S_w) \) is the hydrocarbon pore volume and \( p \) is the oil phase pressure.

The pore volume weighted average pressure is defined as:

\[ FPRP = \frac{\sum PV \cdot p}{\sum PV} \] (Eq. 2.2)

The pore volumes used are those at reference conditions (that is uncompacted pore volumes).

The potential (or “depth corrected pressure”) average calculations are detailed in “Potential Calculations” in the ECLIPSE Technical Description.

ECLIPSE 100

The \( FPRGZ \) and \( RPRGZ \) keywords calculate the P/Z according to:

\[ \frac{p}{Z} = \frac{\sum \left( PV \cdot \frac{S_g}{B_g} \right) \cdot \frac{T}{T_{sc}} \cdot \frac{P_{sc}}{B_g}}{\sum \left( PV \cdot \frac{S_g}{B_g} \right)} \] (Eq. 2.3)

where \( S_g = \) gas saturation
\( B_g = \) gas formation volume factor

Data File Overview
\[ T = \text{reservoir temperature (keyword RTEMP)} \]
\[ T_{sc} = \text{temperature at standard conditions (288K=520R)} \]
\[ \text{P}_{sc} = \text{pressure at standard conditions (14.7psia)} \]

**Equivalent directional keywords for relative permeabilities (ECLIPSE 100)**

*ECLIPSE 100*

If any of the following options are active:

- Directional \( K_r \) (‘DIRECT’ in RUNSPEC keyword SATOPTS)
- Directional End-Point Scaling (‘DIRECT’ in RUNSPEC keyword ENDScale)
- Vertical Equilibrium (RUNSPEC keyword VE)
- Two-point Upstreaming for miscible option (‘TWOPOINT’ in RUNSPEC keyword MISCIBLE),

the following keywords may be used:

<table>
<thead>
<tr>
<th>E100</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>BOKRX</td>
<td>Oil relative permeability in the X direction.</td>
</tr>
<tr>
<td>x</td>
<td>BOKRX-</td>
<td>Oil relative permeability in the -X direction.</td>
</tr>
<tr>
<td>x</td>
<td>BOKRY</td>
<td>Oil relative permeability in the Y direction.</td>
</tr>
<tr>
<td>x</td>
<td>BOKRY-</td>
<td>Oil relative permeability in the -Y direction.</td>
</tr>
<tr>
<td>x</td>
<td>BOKRZ</td>
<td>Oil relative permeability in the Z direction.</td>
</tr>
<tr>
<td>x</td>
<td>BOKRZ-</td>
<td>Oil relative permeability in the -Z direction.</td>
</tr>
<tr>
<td>x</td>
<td>BWKRX</td>
<td>Water relative permeability in the X direction.</td>
</tr>
<tr>
<td>x</td>
<td>BWKRX-</td>
<td>Water relative permeability in the -X direction.</td>
</tr>
<tr>
<td>x</td>
<td>BWKRY</td>
<td>Water relative permeability in the Y direction.</td>
</tr>
<tr>
<td>x</td>
<td>BWKRY-</td>
<td>Water relative permeability in the -Y direction.</td>
</tr>
<tr>
<td>x</td>
<td>BWKRZ</td>
<td>Water relative permeability in the Z direction.</td>
</tr>
<tr>
<td>x</td>
<td>BWKRZ-</td>
<td>Water relative permeability in the -Z direction.</td>
</tr>
<tr>
<td>x</td>
<td>BGKRX</td>
<td>Gas relative permeability in the X direction.</td>
</tr>
<tr>
<td>x</td>
<td>BGKRX-</td>
<td>Gas relative permeability in the -X direction.</td>
</tr>
<tr>
<td>x</td>
<td>BGKRY</td>
<td>Gas relative permeability in the Y direction.</td>
</tr>
<tr>
<td>x</td>
<td>BGKRY-</td>
<td>Gas relative permeability in the -Y direction.</td>
</tr>
<tr>
<td>x</td>
<td>BGKRZ</td>
<td>Gas relative permeability in the Z direction.</td>
</tr>
<tr>
<td>x</td>
<td>BGKRZ-</td>
<td>Gas relative permeability in the -Z direction.</td>
</tr>
<tr>
<td>x</td>
<td>BOKRI</td>
<td>Oil relative permeability in the I direction.</td>
</tr>
<tr>
<td>x</td>
<td>BOKRI-</td>
<td>Oil relative permeability in the -I direction.</td>
</tr>
<tr>
<td>x</td>
<td>BOKRJ</td>
<td>Oil relative permeability in the J direction.</td>
</tr>
<tr>
<td>x</td>
<td>BOKRJ-</td>
<td>Oil relative permeability in the -J direction.</td>
</tr>
<tr>
<td>x</td>
<td>BOKRK</td>
<td>Oil relative permeability in the K direction.</td>
</tr>
</tbody>
</table>
**Table 2.38: Block directional relative permeability quantities (ECLIPSE 100)**

**Processed scaled end-points**

Refer to *"Output of scaled end-points"* in the ECLIPSE Technical Description.

The processed scaled end-point block quantities are:

<table>
<thead>
<tr>
<th>E300</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>BPSWL</td>
<td>Processed scaled connate water saturation</td>
</tr>
<tr>
<td>x</td>
<td>BPSWCR</td>
<td>Processed scaled critical water saturation</td>
</tr>
<tr>
<td>x</td>
<td>BPSWU</td>
<td>Processed scaled maximum water saturation</td>
</tr>
<tr>
<td>x</td>
<td>BPSGL</td>
<td>Processed scaled connate gas saturation</td>
</tr>
</tbody>
</table>
Table 2.39: Processed scaled end-points

Reservoir volumes

The reservoir volume options are:

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>FRPV</td>
<td>RRPV</td>
<td>BRPV</td>
<td>Pore Volume at Reservoir conditions</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BPORV</td>
<td>Cell Pore Volumes at Reference conditions</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FOPV</td>
<td>ROPV</td>
<td>BOPV</td>
<td>Pore Volume containing Oil</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FWPV</td>
<td>RWPV</td>
<td>BWPV</td>
<td>Pore Volume containing Water</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FGPV</td>
<td>RGPV</td>
<td>BGPV</td>
<td>Pore Volume containing Gas</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FHPV</td>
<td>RHPV</td>
<td></td>
<td>Pore Volume containing Hydrocarbon (or “Hydrocarbon Pore Volume”)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BHPV</td>
<td>Pore Volume containing Hydrocarbon (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FRTM</td>
<td>RRTM</td>
<td>BRTM</td>
<td>Transmissibility Multiplier associated with rock compaction</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>BPERMOD</td>
<td>Transmissibility Multiplier associated with rock compaction (as above) If directional transmissibility multipliers are defined, this mnemonic (or BRTM in ECLIPSE 100) is equivalent to BPERMMDX.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>BPERMMDX</td>
<td>Directional Transmissibility Multipliers in the x, y and z directions respectively, associated with rock compaction. These keywords are available when RKTRMDIR is defined in both ECLIPSE 100 and ECLIPSE 300, or when ROCKTRMX, ROCKTRMY and ROCKTRMZ are defined in ECLIPSE 300.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
<td>BPORVMOD</td>
<td>Pore Volume Multiplier associated with rock compaction</td>
</tr>
</tbody>
</table>
### Oil recovery efficiencies

The oil recovery options are:

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Region</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>FOE</td>
<td>(OIP(initial) - OIP(now)) / OIP(initial)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>FOEW</td>
<td>Oil Production from Wells / OIP(initial)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>FOEIW</td>
<td>(OIP(initial) - OIP(now)) / Initial Mobile Oil with respect to Water</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>FOEWW</td>
<td>Oil Production from Wells / Initial Mobile Oil with respect to Water</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>FOEIG</td>
<td>(OIP(initial) - OIP(now)) / Initial Mobile Oil with respect to Gas</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>x</td>
<td>FOEWG</td>
<td>Oil Production from Wells / Initial Mobile Oil with respect to Gas</td>
</tr>
</tbody>
</table>

### Oil recovery mechanism

The oil recovery mechanism options are:

<table>
<thead>
<tr>
<th>E100</th>
<th>Field</th>
<th>Region</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>FORMR</td>
<td>RORMR</td>
<td>Total stock tank oil produced by rock compaction</td>
</tr>
<tr>
<td>x</td>
<td>FORMW</td>
<td>RORMW</td>
<td>Total stock tank oil produced by water influx</td>
</tr>
<tr>
<td>x</td>
<td>FORMG</td>
<td>RORMG</td>
<td>Total stock tank oil produced by gas influx</td>
</tr>
<tr>
<td>x</td>
<td>FORME</td>
<td>RORME</td>
<td>Total stock tank oil produced by oil expansion</td>
</tr>
<tr>
<td>x</td>
<td>FORMS</td>
<td>RORMS</td>
<td>Total stock tank oil produced by solution gas</td>
</tr>
<tr>
<td>x</td>
<td>FORMF</td>
<td>RORMF</td>
<td>Total stock tank oil produced by free gas influx</td>
</tr>
<tr>
<td>x</td>
<td>FORMX</td>
<td>RORMX</td>
<td>Total stock tank oil produced by 'traced' water influx</td>
</tr>
<tr>
<td>x</td>
<td>FORMY</td>
<td>RORMY</td>
<td>Total stock tank oil produced by other water influx</td>
</tr>
<tr>
<td>x</td>
<td>FORFR</td>
<td>RORFR</td>
<td>Fraction of total oil produced by rock compaction</td>
</tr>
<tr>
<td>x</td>
<td>FORFW</td>
<td>RORFW</td>
<td>Fraction of total oil produced by water influx</td>
</tr>
<tr>
<td>x</td>
<td>FORFG</td>
<td>RORFG</td>
<td>Fraction of total oil produced by gas influx</td>
</tr>
<tr>
<td>x</td>
<td>FORFE</td>
<td>RORFE</td>
<td>Fraction of total oil produced by oil expansion</td>
</tr>
<tr>
<td>x</td>
<td>FORFS</td>
<td>RORFS</td>
<td>Fraction of total oil produced by solution gas</td>
</tr>
<tr>
<td>x</td>
<td>FORFF</td>
<td>RORFP</td>
<td>Fraction of total oil produced by free gas influx</td>
</tr>
<tr>
<td>x</td>
<td>FORFX</td>
<td>RORFX</td>
<td>Fraction of total oil produced by 'traced' water influx</td>
</tr>
</tbody>
</table>
Table 2.42: Oil recovery mechanism (ECLIPSE 100)

<table>
<thead>
<tr>
<th>Field</th>
<th>Region</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x FORFY</td>
<td>RORFY</td>
<td>Fraction of total oil produced by other water influx</td>
</tr>
</tbody>
</table>

**Note:** The oil recovery mechanism keywords are not available in runs with vaporized oil present.

See also "Recovery Mechanisms" in the *ECLIPSE Technical Description*.

### Recovery plant options (ECLIPSE 300 only)

The recovery plant options are given in the table below. Notes for this table:

1. See “Format of SUMMARY keywords with a component index”.

<table>
<thead>
<tr>
<th>Field</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>FNRR</td>
<td>NGL recovery volume rate</td>
</tr>
<tr>
<td>FNRT</td>
<td>NGL recovery volume total</td>
</tr>
<tr>
<td>FNRM</td>
<td>NGL recovery molar rate</td>
</tr>
<tr>
<td>FGRR</td>
<td>Gas remaining volume rate</td>
</tr>
<tr>
<td>FGRT</td>
<td>Gas remaining volume total</td>
</tr>
<tr>
<td>FGRM</td>
<td>Gas remaining molar rate</td>
</tr>
<tr>
<td>FXNR</td>
<td>NGL recovery composition. See Note 1.</td>
</tr>
<tr>
<td>FYGR</td>
<td>Gas remaining composition. See Note 1.</td>
</tr>
</tbody>
</table>

*Table 2.43: Keywords to obtain reports of recovered NGLs and gas (ECLIPSE 300 only)*

### Analytic aquifer quantities

The analytic aquifer options are given in the table below. Notes for this table:

1. See “Format of SUMMARY keywords with a component index”.

<table>
<thead>
<tr>
<th>Field</th>
<th>Aquifer</th>
<th>Aquifer List</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x x FAAQR</td>
<td>AAQR</td>
<td>Aquifer influx rate (for liquid aquifer)</td>
<td></td>
</tr>
<tr>
<td>x FAAQT</td>
<td>AAQT</td>
<td>Cumulative aquifer influx (for liquid aquifer)</td>
<td></td>
</tr>
<tr>
<td>x FAAQG</td>
<td>AAQRG</td>
<td>Aquifer influx rate (for gas aquifer)</td>
<td></td>
</tr>
<tr>
<td>x FAAQG</td>
<td>ALQG</td>
<td>Aquifer influx rate (for gas aquifer) (as above)</td>
<td></td>
</tr>
<tr>
<td>x FAAQT</td>
<td>AAQTG</td>
<td>Cumulative aquifer influx (for gas aquifer)</td>
<td></td>
</tr>
<tr>
<td>x FAAQG</td>
<td>ALQTG</td>
<td>Cumulative aquifer influx (for gas aquifer) (as above)</td>
<td></td>
</tr>
<tr>
<td>x AACMR</td>
<td>Aquifer component molar influx rate (for multi-component aquifer). See Note 1.</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Data File Overview
### Numerical aquifer quantities

The numerical aquifer options given in the table below. Notes for this table:

1. See “Format of SUMMARY keywords with a component index”.

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Aquifer</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>AACMT</td>
<td>Aquifer component molar influx totals (for multi-</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>component aquifer). See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td>AAQP</td>
<td>Aquifer pressure (excludes constant flux aquifer as</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>given by AQUFLUX).</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FAQER</td>
<td>AAQER</td>
<td>Aquifer thermal energy influx rate (Thermal option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FAQET</td>
<td>AAQET</td>
<td>Cumulative aquifer thermal energy influx (Thermal</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>AAQTEMP</td>
<td>Aquifer temperature</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>AAQENTH</td>
<td>Aquifer molar enthalpy (Thermal option)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td>AAQTD</td>
<td>Aquifer dimensionless time (Carter-Tracy aquifers only)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td></td>
<td>AAQPD</td>
<td>Aquifer dimensionless pressure (Carter-Tracy aquifers only)</td>
</tr>
</tbody>
</table>

*Table 2.44: Analytic aquifer quantities*

### Tracer and API tracking quantities

The tracer and API tracking options are:

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>FTPR</td>
<td>GTPR</td>
<td>WTPR</td>
<td>CTFR</td>
<td></td>
<td></td>
<td>Tracer Flow Rate (+ or -)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FTPR</td>
<td>GTPT</td>
<td>WTPR</td>
<td>CTPT</td>
<td></td>
<td></td>
<td>Tracer Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FTPC</td>
<td>GTPC</td>
<td>WTPC</td>
<td>CTPC</td>
<td></td>
<td></td>
<td>Tracer Production Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FTPC</td>
<td>GTIR</td>
<td>WTPC</td>
<td>CTIR</td>
<td></td>
<td></td>
<td>Tracer Production Concentration</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FTPC</td>
<td>GTIT</td>
<td>WTPT</td>
<td>CTIT</td>
<td></td>
<td></td>
<td>Tracer Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FTPC</td>
<td>GTIC</td>
<td>WTPC</td>
<td>CTIC</td>
<td></td>
<td></td>
<td>Tracer Injection Concentration</td>
</tr>
</tbody>
</table>

*Table 2.45: Numerical aquifer quantities*
Table 2.46: Tracer and API tracking quantities

Keywords which belong specifically to the Tracer Tracking option (that is with T as the second letter in the list above) should be compounded with the name of the tracer to which they refer. For example, if the field production rate of a tracer called OL1 is required on the Summary files, then the keyword FTPROL1 should be used.

**Brine option quantities**

The brine options are:

---

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Tracer inter-region Flow Total (Free)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Tracer inter-region Flow Total (Solution)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Tracer inter-region Flow Total (Total)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Oil API (for API tracking). (F,G,W,C)API are for production wells</td>
</tr>
</tbody>
</table>

---

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Salt Flow Rate (+ or -)</td>
</tr>
</tbody>
</table>
Table 2.47: Brine option quantities

The outputs for the Multi-Component Brine are:

<table>
<thead>
<tr>
<th>E100</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FTPRANI</td>
<td>GTPRANI</td>
<td>WTPRANI</td>
<td></td>
<td>CTFRANI</td>
<td>Anion Flow Rate</td>
</tr>
<tr>
<td>x</td>
<td>FTPTANI</td>
<td>GTPTANI</td>
<td>WTPTANI</td>
<td>CPTANI</td>
<td></td>
<td>Anion Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>FTIRANI</td>
<td>GTIRANI</td>
<td>WTIRANI</td>
<td></td>
<td></td>
<td>Anion Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>FTITANI</td>
<td>GTITANI</td>
<td>WTITANI</td>
<td>CTITANI</td>
<td></td>
<td>Anion Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>FTPRCAT</td>
<td>GTPRCAT</td>
<td>WTPRCAT</td>
<td></td>
<td>CTFRCAT</td>
<td>Cation Flow Rate</td>
</tr>
<tr>
<td>x</td>
<td>FTPTCAT</td>
<td>GTPTCAT</td>
<td>WTPTCAT</td>
<td>CPTCAT</td>
<td></td>
<td>Cation Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>FTIRCAT</td>
<td>GTIRCAT</td>
<td>WTIRCAT</td>
<td></td>
<td></td>
<td>Cation Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>FTITCAT</td>
<td>GTITCAT</td>
<td>WTITCAT</td>
<td>CTITCAT</td>
<td></td>
<td>Cation Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>FTPRCAT</td>
<td>GTPRCAT</td>
<td>WTPRCAT</td>
<td></td>
<td>BTCNFANI</td>
<td>Anion Flowing Concentration</td>
</tr>
<tr>
<td>x</td>
<td>FTPTCAT</td>
<td>GTPTCAT</td>
<td>WTPTCAT</td>
<td>CPTCAT</td>
<td></td>
<td>Cation Flow Rate</td>
</tr>
<tr>
<td>x</td>
<td>FTIRCAT</td>
<td>GTIRCAT</td>
<td>WTIRCAT</td>
<td></td>
<td></td>
<td>Cation Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>FTITCAT</td>
<td>GTITCAT</td>
<td>WTITCAT</td>
<td>CTITCAT</td>
<td></td>
<td>Cation Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>FTPRCAT</td>
<td>GTPRCAT</td>
<td>WTPRCAT</td>
<td></td>
<td>BTCNFCAT</td>
<td>Cation Flowing Concentration</td>
</tr>
<tr>
<td>x</td>
<td>BTRADCAT</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Cation Rock Associated Concentration</td>
</tr>
<tr>
<td>x</td>
<td>BTSADCAT</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Cation Surfactant Associated Concentration</td>
</tr>
<tr>
<td>x</td>
<td>BESALSR</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Effective Salinity with respect to Surfactant</td>
</tr>
</tbody>
</table>

Effective water viscosity due to salt concentration. Does not apply if the Polymer option is active; in that case, this vector will always be zero.
**Temperature option quantities**

Information about the temperatures in the reservoir can be output to the Summary file in an analogous manner to the tracer output. For the heat equation the three-character mnemonic that should be appended to summary quantities is ‘HEA’.

<table>
<thead>
<tr>
<th>E100</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Region</th>
<th>Block</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>FTPC</td>
<td>GTPC</td>
<td>WTPC</td>
<td></td>
<td>BESALPLY</td>
<td>Effective Salinity art Polymer</td>
</tr>
</tbody>
</table>

**Table 2.48: Outputs for Multi-Component Brine (ECLIPSE 100)**

**Environmental tracers**

The following keywords control output of data specific to tracers:

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>FTPR</td>
<td>GTPR</td>
<td>WTPR</td>
<td>CTFR</td>
<td></td>
<td></td>
<td>Tracer Flow Rate (+ or -)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FTPR</td>
<td>GTPR</td>
<td>WTPR</td>
<td>CTPR</td>
<td></td>
<td></td>
<td>Tracer Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FTPT</td>
<td>GTPT</td>
<td>WTPT</td>
<td>CTPT</td>
<td></td>
<td></td>
<td>Tracer Production Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FTPC</td>
<td>GTPC</td>
<td>WTPC</td>
<td>CTPC</td>
<td></td>
<td></td>
<td>Tracer Production Concentration</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FTIR</td>
<td>GTIR</td>
<td>WTIR</td>
<td>CTIR</td>
<td></td>
<td></td>
<td>Tracer Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FTIT</td>
<td>GTIT</td>
<td>WTIT</td>
<td>CTIT</td>
<td></td>
<td></td>
<td>Tracer Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FTIC</td>
<td>GTIC</td>
<td>WTIC</td>
<td>CTIC</td>
<td></td>
<td></td>
<td>Tracer Injection Concentration</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BTCNF</td>
<td></td>
<td></td>
<td>Tracer Concentration (Free)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BTCNS</td>
<td></td>
<td></td>
<td>Tracer Concentration (Solution)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BTCN#</td>
<td></td>
<td></td>
<td>Tracer concentration in phase # (1,2,3,...)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTPIPT</td>
<td></td>
<td></td>
<td>RTIPT</td>
<td></td>
<td></td>
<td>Tracer In Place (Total)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTPIFF</td>
<td></td>
<td></td>
<td>RTIPF</td>
<td></td>
<td></td>
<td>Tracer In Place (Free)</td>
</tr>
</tbody>
</table>

**Table 2.49: Summary output for Temperature option (ECLIPSE 100)**
<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td>FTIPS</td>
<td></td>
<td></td>
<td>RTIPS</td>
<td></td>
<td>BTIPS</td>
<td>Tracer In Place (Solution)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTIP#</td>
<td></td>
<td></td>
<td>RTIP#</td>
<td></td>
<td>BTIP#</td>
<td>Tracer In Place in phase # (1,2,3,...)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>RTFTT</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Tracer inter-region Flow (Total)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>RTFTF</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Tracer inter-region Flow (Free)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>RTFTS</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Tracer inter-region Flow (Solution)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>RTFT#</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Tracer inter-region Flow in phase # (1,2,3,...)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTADS</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BTADS</td>
<td>Tracer Adsorption total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Tracer Adsorption</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTDCY</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>RTDCY</td>
<td>Decayed tracer</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTIRF</td>
<td>GTIRF</td>
<td>WTIHF</td>
<td>CTIRF</td>
<td></td>
<td></td>
<td>Tracer Injection Rate (Free)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTIRS</td>
<td>GTIRS</td>
<td>WTIJS</td>
<td>CTIRS</td>
<td></td>
<td></td>
<td>Tracer Injection Rate (Solution)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTPRF</td>
<td>GTPRF</td>
<td>WTPRF</td>
<td>CTPRF</td>
<td></td>
<td></td>
<td>Tracer Production Rate (Free)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTPRS</td>
<td>GTPRS</td>
<td>WTPRS</td>
<td>CTPRS</td>
<td></td>
<td></td>
<td>Tracer Production Rate (Solution)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTITF</td>
<td>GTITF</td>
<td>WITTF</td>
<td>CITTF</td>
<td></td>
<td></td>
<td>Tracer Injection Total (Free)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTITS</td>
<td>GTITS</td>
<td>WITIS</td>
<td>CITIS</td>
<td></td>
<td></td>
<td>Tracer Injection Total (Solution)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTPTF</td>
<td>GTPTF</td>
<td>WTPTF</td>
<td>CTPTF</td>
<td></td>
<td></td>
<td>Tracer Production Total (Free)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTPTS</td>
<td>GTPTS</td>
<td>WTPTS</td>
<td>CTPTS</td>
<td></td>
<td></td>
<td>Tracer Production Total (Solution)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTICF</td>
<td>GTICF</td>
<td>WTICF</td>
<td>CTICF</td>
<td></td>
<td></td>
<td>Tracer Injection Concentration (Free)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTICS</td>
<td>GTICS</td>
<td>WTICS</td>
<td>CTICS</td>
<td></td>
<td></td>
<td>Tracer Injection Concentration (Solution)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTPCF</td>
<td>GTPCF</td>
<td>WTPCF</td>
<td>CTPCF</td>
<td></td>
<td></td>
<td>Tracer Production Concentration (Free)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTPCS</td>
<td>GTPCS</td>
<td>WTPCS</td>
<td>CTPCS</td>
<td></td>
<td></td>
<td>Tracer Production Concentration (Solution)</td>
</tr>
</tbody>
</table>

*Table 2.50: Environmental tracer SUMMARY output controls*

The keywords above should be compounded with the name of the tracer to which they refer. For example, if the field production rate of a tracer called OL1 is required on the Summary files, then the keyword FTPROL1 should be used in the SUMMARY section.

For partitioned tracers, the concentration, flow, and amount in place in a specific phase can be output by using either the F (Free) and S (Solution) specifiers, or by using the phase number #. For a tracer partitioning into the components oil, water and gas, the F and S specifiers are no longer sufficient to
uniquely determine the phase of interest. In this case the phase number must be used. For example, BTCN3OL1 refers to the block concentration of tracer OL1 in phase 3 as specified in the list of phases when defining the tracer partitioning functions (see keyword TRACERKM). If the S (Solution) specifier is still used for a tracer partitioning into 3 or more phases, the concentration output will be the concentration in the last phase as defined in TRACERKM.

Note: BTADS reports the amount of adsorbed tracer per unit mass of rock in a given block (for example ‘kg/kg’). The corresponding region and field keywords, RTADS and FTADS, report the total amount of adsorbed tracer (for example ‘kg’).

**Coal Bed Methane Model (ECLIPSE 100)**

This model is for ECLIPSE 100 only. The following keywords control output of data specific to the Coal bed Methane Model:

<table>
<thead>
<tr>
<th>E100</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>FMPR</td>
<td>GMPR</td>
<td>WMPR</td>
<td></td>
<td></td>
<td>Methane Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>FMPT</td>
<td>GMPT</td>
<td>WMPT</td>
<td></td>
<td></td>
<td>Methane Production Total</td>
</tr>
<tr>
<td>x</td>
<td>FMIR</td>
<td>GMIR</td>
<td>WMIR</td>
<td></td>
<td></td>
<td>Methane Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>FMIT</td>
<td>GMIT</td>
<td>WMIT</td>
<td></td>
<td></td>
<td>Methane Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>FCGC</td>
<td></td>
<td></td>
<td>RCGC</td>
<td>BCGC</td>
<td>Bulk Coal Gas Concentration (concentration in the coal matrix)</td>
</tr>
<tr>
<td>x</td>
<td>FCSC</td>
<td></td>
<td></td>
<td>RCSC</td>
<td>BCSC</td>
<td>Bulk Coal Solvent Concentration (concentration in the coal matrix)</td>
</tr>
</tbody>
</table>

Table 2.51: Coal Bed Methane (ECLIPSE 100)

Note: Methane production is identical to the gas production unless the solvent option is active. In that case, the gas production rate is the total gas + solvent production rate.

**Foam Model**

The following keywords control output of data specific to the Foam Model.

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td>FTPRFOA</td>
<td>GTPRFOA</td>
<td>WTPRFOA</td>
<td>CTRFROA</td>
<td>Flow Rate (+ or -)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTPTFOA</td>
<td>GPTTFOA</td>
<td>WTPTFOA</td>
<td>CTPTFOA</td>
<td>Production Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTIRFOA</td>
<td>GTRFOA</td>
<td>WTRFOA</td>
<td>CTRFOA</td>
<td>Production Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTITFOA</td>
<td>GTITFOA</td>
<td>WTITFOA</td>
<td>CTITFOA</td>
<td>Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTIPTFOA</td>
<td></td>
<td></td>
<td></td>
<td>Injection Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTDFOA</td>
<td></td>
<td></td>
<td></td>
<td>In Solution</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTDFOA</td>
<td></td>
<td></td>
<td></td>
<td>Adsorption total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>FTDFOA</td>
<td></td>
<td></td>
<td></td>
<td>Decayed tracer</td>
</tr>
</tbody>
</table>

Data File Overview
Table 2.52: Foam SUMMARY field, group, well and connection output keywords

<table>
<thead>
<tr>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x FTMOBFOA</td>
<td></td>
<td></td>
<td></td>
<td>Gas mobility factor (excluding shear)</td>
</tr>
</tbody>
</table>

Table 2.53: Foam SUMMARY, field, region and block output keywords

<table>
<thead>
<tr>
<th>Field</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>FTPRFOA</td>
<td></td>
<td></td>
<td>Production Rate</td>
</tr>
<tr>
<td>FTPTFOA</td>
<td></td>
<td></td>
<td>Production Total</td>
</tr>
<tr>
<td>FTIRFOA</td>
<td></td>
<td></td>
<td>Injection Rate</td>
</tr>
<tr>
<td>FTITFOA</td>
<td></td>
<td></td>
<td>Injection Total</td>
</tr>
<tr>
<td>BTCNFFOA</td>
<td></td>
<td></td>
<td>Concentration</td>
</tr>
<tr>
<td>BFOAM</td>
<td></td>
<td></td>
<td>Surfactant concentration</td>
</tr>
<tr>
<td>BTCNMFOA</td>
<td></td>
<td></td>
<td>Capillary number</td>
</tr>
<tr>
<td>BFOAMCNM</td>
<td></td>
<td></td>
<td>Capillary number</td>
</tr>
<tr>
<td>FTIPTFOA</td>
<td>RTIPTFOA</td>
<td></td>
<td>In Solution</td>
</tr>
<tr>
<td>RTFTTFOA</td>
<td></td>
<td></td>
<td>Inter-region Flow Total</td>
</tr>
<tr>
<td>FTADSFOA</td>
<td>RTADSFOA</td>
<td></td>
<td>Adsorption total</td>
</tr>
<tr>
<td>BTADSFOA</td>
<td></td>
<td></td>
<td>Adsorption</td>
</tr>
<tr>
<td>FTDCYFOA</td>
<td>RTDCYFOA</td>
<td></td>
<td>Decayed tracer</td>
</tr>
<tr>
<td>BTDCYFOA</td>
<td></td>
<td></td>
<td>Gas mobility factor (excluding shear)</td>
</tr>
<tr>
<td>BFOAMMOB</td>
<td></td>
<td></td>
<td>Gas mobility factor</td>
</tr>
<tr>
<td>BTHLFFOA</td>
<td></td>
<td></td>
<td>Decay Half life</td>
</tr>
<tr>
<td>BFOAMST</td>
<td></td>
<td></td>
<td>Gas-water surface tension</td>
</tr>
</tbody>
</table>

Gas Field Operations Model

The following items are relevant to the Gas Field Operations Model:

<table>
<thead>
<tr>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x x FSGR</td>
<td>GSGR</td>
<td></td>
<td>Sales Gas Rate</td>
</tr>
<tr>
<td>x x FGSR</td>
<td>GGSR</td>
<td></td>
<td>Sales Gas Rate (as above)</td>
</tr>
<tr>
<td>x x FSGT</td>
<td>GSGT</td>
<td></td>
<td>Sales Gas Total</td>
</tr>
<tr>
<td>x x FGST</td>
<td>GGST</td>
<td></td>
<td>Sales Gas Total (as above)</td>
</tr>
<tr>
<td>x x FGDC</td>
<td>GGDC</td>
<td>WGDC</td>
<td>Gas Delivery Capacity</td>
</tr>
<tr>
<td>x x FGDCQ</td>
<td>GGDCQ</td>
<td></td>
<td>Field/Group Gas DCQ</td>
</tr>
<tr>
<td>x</td>
<td>GMCPL</td>
<td></td>
<td>Group Multi-level Compressor Level</td>
</tr>
<tr>
<td>x x GPR</td>
<td></td>
<td></td>
<td>Group nodal PRessure in network When using the Extended Network Model, the names must correspond to network nodes instead of groups</td>
</tr>
</tbody>
</table>
Table 2.54: Gas field operations SUMMARY output controls

The mnemonics FGPR, GGPR, FGPT, GGPT refer to the gross gas produced from the formation, without allowing for consumption. GGDC and WGDC refer to the group and well gas sales rates when the field is operating at its delivery capacity. GGDCQ should be used only for contract groups in cases where two or more groups have separate gas supply contracts (see keyword GSWINGF). FGDCQ should be used when there is a single contract applied to the FIELD (keyword SWINGFAC). GPR provides the group nodal pressures when there is a production network. GPRDC may be used to obtain the group nodal pressures that exist then the field is operating at delivery capacity. When using the Extended Network Model, the names must correspond to network nodes instead of groups. The mnemonics WGPRFP, WTHPPF, WBHPFP, GPRFP, GGPRNBFP extract data from the first pass of the Gas field operations Model. However, these mnemonics do not refer to continuous quantities but rather the values of these quantities at the end of the first pass. Inclusion of any of these mnemonics in the summary section will force an additional iteration of the first pass in order that converged data is recorded.

**Simulator performance keyword**

NGOPAS Outputs the number of iterations used to converge the DCQ in the first pass of the contract period.

**Gas lift optimization facility**

The following keywords control output of data specific to the Gas Lift Optimization facility:
Gas Lift Injection Rate (Assumes ALQ = gas lift injection rate. Well rates are multiplied by their efficiency factors when summing up group and field rates.)

Well Oil Gas Lift Ratio (This is the incremental change in the field’s production rate with respect to the addition of another gas lift increment to that well. The denominator in this expression may optionally include a weighted version of the incremental change in gas production rate.)

Table 2.55: Gas lift optimization SUMMARY output controls

Note: In the presence of gas lift optimization, the produced gas reported (for example using the FGPR, GGPR and WGPR keywords) does not include produced lift gas, only produced formation gas.

Gas calorific value control option

The following items are relevant to the Gas Calorific Value Control option:

Table 2.56: Gas calorific value control SUMMARY output controls

The energy production rates and totals (FEPR, GEPR, FEPT, GEPT) refer to the gross energy produced from the formation, without allowing for consumption. The energy sales rates and totals (FESR, GESR, FEST, GEST) refer to the energy produced, minus the energy injected, minus the energy of any consumed gas. The energy delivery capacity and DCQ are available only when using the Gas Field Operations Model. GEDC and WEDC refer to the group and well energy sales rates when the field is operating at its delivery capacity. GEDCQ should be used only for contract groups in cases where two or more groups have separate gas supply contracts (see keyword GSWINGF). FEDCQ should be used when there is a single contract applied to the FIELD (keyword SWINGFAC).

Gi pseudo-compositional model

ECLIPSE 100 only

The following keywords control output of data specific to the Gi Model:

Table 2.57: Gi pseudo-compositional SUMMARY output controls

Data File Overview
Local grid refinement and coarsening

In ECLIPSE 100 LGR runs, two sets of summary data are output. The usual Summary files contain summary data produced at the global timesteps. In addition summary file quantities are output at the local timesteps; this local data is output to a separate LGR file.

The local timestep summary data is stored in a file with the following name:

- ROOT.FLGR (or ROOT.FLG)
  Formatted
- ROOT.LGR
  Unformatted

The creation of local timestep Summary files can be prevented by using the NOSUMLGR keyword in the SUMMARY section. For example, when running lgr-in-place there is little point in dumping Summary files for the local timesteps since the global and local timesteps are all synchronized.

Due to the local timestepping used on grid refinements in ECLIPSE 100, it is possible to request global grid summary quantities for wells that are defined on a local grid. This is not possible in ECLIPSE 300.

Summary data for the local grid is requested by using SUMMARY keywords with the first letter L. Three types of keyword are available:

- LB* Local Block data
- LC* Local Connection data
- LW* Local Well data

The keywords available are the same as the global keyword with the ‘L’ added to the front. For example LWBHP requested the WBHP for the local well.

Note that the maximum number of characters in a SUMMARY keyword is 8. This may introduce limitations in very specific cases for example in tracer block quantities.

Keywords that begin with LB are described in "Format of SUMMARY keywords beginning with LB".
Keywords that begin with LC are described in "Format of SUMMARY keywords beginning with LC".
Keywords that begin with LW are described in "Format of SUMMARY keywords beginning with LW".

Operational keywords

NOSUMLGR Prevents the creation of local timestep summary files.

Plotting local timestep results in GRAF

When loading the data into GRAF, you are prompted to say whether the local timestep data should be loaded along with the usual summary files. If the local timestep data (from the ROOT.LGR file) is loaded, a set of vectors is created for each local grid. These sets of vectors (with their own time vector) are identified by having a different ‘origin’.

You are prompted for the origin for each local grid. However these origins can be defaulted to an 8 character string of the form:

- AAA_BBBB
where AAA is the first three characters of the root name for the run and 
BBBB is the first four characters of the LGR name.

For example, run ROOT containing LGRs called SOUTH and NORTH would produce sets of vectors with origins ROO_SOUT and ROO_NORT.

Note: This means it is useful to give LGRs names that differ in the first 4 characters.

The global timestep data is treated in the usual way in GRAF.

Multisegment wells

The keywords controlling output of segment data specific to multisegment wells are given in the table below. Notes for this table:

1. See “Format of SUMMARY keywords with a component index”.

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Segment</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>SOFR</td>
<td>Segment Oil Flow Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SOFRF</td>
<td>Segment Free Oil Flow Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SOFRS</td>
<td>Segment Solution Oil Flow Rate (vaporized oil)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SWFR</td>
<td>Segment Water Flow Rate</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SGFR</td>
<td>Segment Gas Flow Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SGFRF</td>
<td>Segment Free Gas Flow Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SGFRS</td>
<td>Segment Solution Gas Flow Rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SKFR</td>
<td>Segment Component Flow Rate. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SCWGFR</td>
<td>Segment Component Flow Rate as Wet Gas. See Note 1.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SHFR</td>
<td>Segment Enthalpy Flow Rate (Thermal option)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SWCT</td>
<td>Segment Water Cut</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SGOR</td>
<td>Segment Gas Oil Ratio</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SOGR</td>
<td>Segment Oil Gas Ratio</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SWGR</td>
<td>Segment Water Gas Ratio</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SPR</td>
<td>Segment PRessure</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SPRD</td>
<td>Segment PRessure Drop</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SPRDF</td>
<td>Segment PRessure Drop component due to Friction</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SPRDH</td>
<td>Segment PRessure Drop component due to Hydrostatic head</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SPRDA</td>
<td>Segment PRessure drop due to Acceleration head</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SPRDM</td>
<td>Segment frictional PRessure Drop Multiplier (keyword WSEGMULT)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SPPOW</td>
<td>Working power of a pull through pump (keyword WSEGPULL)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SOFV</td>
<td>Segment Oil Flow Velocity</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SWFV</td>
<td>Segment Water Flow Velocity</td>
</tr>
<tr>
<td>E100</td>
<td>E300</td>
<td>Segment</td>
<td>Information</td>
</tr>
<tr>
<td>------</td>
<td>------</td>
<td>---------</td>
<td>-------------</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SGFV</td>
<td>Segment Gas Flow Velocity</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SOHF</td>
<td>Segment Oil Holdup Fraction</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SWHF</td>
<td>Segment Water Holdup Fraction</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SGHF</td>
<td>Segment Gas Holdup Fraction</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SDENM</td>
<td>Segment fluid mixture density</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SOVIS</td>
<td>Segment oil viscosity</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SWVIS</td>
<td>Segment water viscosity (pure water)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SGVIS</td>
<td>Segment gas viscosity</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SEMVIS</td>
<td>Segment effective mixture viscosity (water/polymer)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SGLPP</td>
<td>Segment Gas-Liquid Profile Parameter, C0 (drift flux slip model)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SGLVD</td>
<td>Segment Gas-Liquid Drift Velocity, Vd (drift flux slip model)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SOWPP</td>
<td>Segment Oil-Water Profile Parameter, C0 (drift flux slip model)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SOWVD</td>
<td>Segment Oil-Water Drift Velocity, Vd (drift flux slip model)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SOIMR</td>
<td>Segment Oil Import Rate (keyword <strong>WSEGEXSS</strong>)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SGIMR</td>
<td>Segment Gas Import Rate (keyword <strong>WSEGEXSS</strong>)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SWIMR</td>
<td>Segment Water Import Rate (keyword <strong>WSEGEXSS</strong>)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SHIMR</td>
<td>Segment Enthalpy Import Rate (keyword <strong>WSEGEXSS</strong>) (Thermal option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SORMR</td>
<td>Segment Oil Removal Rate (keyword <strong>WSEGEXSS</strong>)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SGRMR</td>
<td>Segment Gas Removal Rate (keyword <strong>WSEGEXSS</strong>)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SWRMR</td>
<td>Segment Water Removal Rate (keyword <strong>WSEGEXSS</strong>)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SHRMR</td>
<td>Segment Enthalpy Removal Rate (keyword <strong>WSEGEXSS</strong>) (Thermal option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SOIMT</td>
<td>Segment Oil Import Total (keyword <strong>WSEGEXSS</strong>)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SGIMT</td>
<td>Segment Gas Import Total (keyword <strong>WSEGEXSS</strong>)</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SWIMT</td>
<td>Segment Water Import Total (keyword <strong>WSEGEXSS</strong>)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SHIMT</td>
<td>Segment Enthalpy Import Total (keyword <strong>WSEGEXSS</strong>) (Thermal option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SORMT</td>
<td>Segment Oil Removal Total (keyword <strong>WSEGEXSS</strong>)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SGRMT</td>
<td>Segment Gas Removal Total (keyword <strong>WSEGEXSS</strong>)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SWRMT</td>
<td>Segment Water Removal Total (keyword <strong>WSEGEXSS</strong>)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SHRMT</td>
<td>Segment Enthalpy Removal Total (keyword <strong>WSEGEXSS</strong>) (Thermal option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SAPI</td>
<td>Segment API value</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SCFR</td>
<td>Segment polymer flow rate (Polymer flood option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SCCN</td>
<td>Segment polymer concentration (Polymer flood option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SSFR</td>
<td>Segment brine flow rate (Surfactant and Polymer flood option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SSCN</td>
<td>Segment brine concentration (Surfactant and Polymer flood option)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>STFR</td>
<td>Segment tracer flow rate</td>
</tr>
</tbody>
</table>
### Table 2.58: Multisegment wells SUMMARY output controls

The segment flow rates reported are the flow rates of each phase through the tubing past the segment’s end nearest the well head. Positive values represent flow towards the well head (production), while negative values represent flow away from the well head (injection). The flow rates refer to the sum of free and dissolved phases at surface conditions, equivalent to the reported well rates. The flow velocities refer to the local flow velocities of the free phases through the segment. The flow velocities of all the phases should be equal unless you are using a built-in Multi-phase Flow Model that allows slip. The holdup fractions refer to the local volume fraction of the segment occupied by each free phase.

For drift-flux segments, the phases can flow in different directions. Therefore phase flow ratios (such as SWCT, SGOR, SOGR and SWGR) may be negative, and the water cut (SWCT) could be infinite if the oil and water flow volumes are the same, but in different directions, so the net liquid flow is zero.

The tracer flow rate and concentration keywords should be compounded with the name of the tracer to which they refer (for example STFROL1 and STFCOL1 give the flow rate and concentration of the tracer OL1).

#### Rivers

The following keywords control output of reach data specific to rivers

<table>
<thead>
<tr>
<th>E100</th>
<th>Segment</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>SRRQR</td>
<td>Reach flow at current time</td>
</tr>
<tr>
<td>x</td>
<td>SRRQT</td>
<td>Reach cumulative flow</td>
</tr>
<tr>
<td>x</td>
<td>SRBQR</td>
<td>Branch flow at current time</td>
</tr>
<tr>
<td>x</td>
<td>SRBQT</td>
<td>Branch cumulative flow</td>
</tr>
</tbody>
</table>
Table 2.59: River SUMMARY output controls

The river reach controls are:

<table>
<thead>
<tr>
<th>E100</th>
<th>Segment</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>SRTQR</td>
<td>River total flow at current time</td>
</tr>
<tr>
<td>x</td>
<td>SRTQT</td>
<td>River total cumulative flow</td>
</tr>
<tr>
<td>x</td>
<td>SRRFLOW</td>
<td>Reach flux through cross-sectional area at current time</td>
</tr>
<tr>
<td>x</td>
<td>SRRAREA</td>
<td>Reach area at current time</td>
</tr>
<tr>
<td>x</td>
<td>SRRDEPTH</td>
<td>Reach depth at current time</td>
</tr>
<tr>
<td>x</td>
<td>SRREXCH</td>
<td>Exchange flux at current time</td>
</tr>
<tr>
<td>x</td>
<td>SRRFRODE</td>
<td>Reach Froude number at current time</td>
</tr>
<tr>
<td>x</td>
<td>SRRHEAD</td>
<td>Reach hydraulic head at current time</td>
</tr>
<tr>
<td>x</td>
<td>SRTFR</td>
<td>Reach tracer flow rate</td>
</tr>
<tr>
<td>x</td>
<td>SRTFC</td>
<td>Reach tracer concentration</td>
</tr>
<tr>
<td>x</td>
<td>SRSFR</td>
<td>Reach brine flow rate through connections</td>
</tr>
<tr>
<td>x</td>
<td>SRSFC</td>
<td>Reach brine concentration</td>
</tr>
</tbody>
</table>

Table 2.60: River reach SUMMARY controls

For example,

```plaintext
SRFFLOW
   -- FLUX FOR REACH 12
   MYRIVER 12 / 
   / 
SRBQR
   -- FLOW FOR BRANCH 1
   MYRIVER 1 / 
   / 
SRTQR
   -- TOTAL FLOW FOR ENTIRE RIVER
   MYRIVER / 
   / 
SRBQT
   -- CUMULATIVE FLOW FOR BRANCH 1
   MYRIVER 1 / 
SRTFF
   -- TRACER 'FFF' FLOW RATE FOR REACH 1
   MYRIVER 1 / 
CRRPROT
   -- CUMULATIVE WATER PRODUCTION FOR BLOCK 11 20 2
   MYRIVER 11 20 2 / 
SRSFR
   -- BRINE FLOW TO REACHES 11 AND 12
   MYRIVER 11 / 
```

ECLIPSE Reference Manual

Data File Overview
**Network option**

The following keywords control output of data specific to the Network option.

When using the Extended Network Model, the list of names following these keywords must correspond to network nodes instead of **GRUPTREE** groups.

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Group</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>GPR</td>
<td>Group or node PRessure in the production network</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GPRG</td>
<td>Group or node PRessure in the gas injection network</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GPRW</td>
<td>Group or node PRessure in the water injection network</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GPRB</td>
<td>Pressure drop along the group’s or node’s outlet branch in the production network</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GPRBG</td>
<td>Pressure drop along the group’s or node’s inlet branch in the gas injection network</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GPRBW</td>
<td>Pressure drop along the group’s or node’s inlet branch in the water injection network</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GALQ</td>
<td>ALQ in the group’s or node’s outlet branch in the production network. No units are given, as there are several possible definitions for this quantity.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GOPRNB</td>
<td>Oil flow rate along the group’s or node’s outlet branch in the production network</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GWPRNB</td>
<td>Water flow rate along the group’s or node’s outlet branch in the production network</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GGPRNB</td>
<td>Gas flow rate along the group’s or node’s outlet branch in the production network</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>GLPRNB</td>
<td>Liquid flow rate along the group’s or node’s outlet branch in the production network</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GWIRNB</td>
<td>Water flow rate along the group’s or node’s inlet branch in the water injection network</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GGIRNB</td>
<td>Gas flow rate along the group’s or node’s inlet branch in the gas injection network</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>GOMNR</td>
<td>Group or node minimum oil rate as specified with <strong>GNETDP</strong> in the production network</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>GGMNR</td>
<td>Group or node minimum gas rate as specified with <strong>GNETDP</strong> in the production network</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>GWMNR</td>
<td>Group or node minimum water rate as specified with <strong>GNETDP</strong> in the production network</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>GLMNR</td>
<td>Group or node minimum liquid rate as specified with <strong>GNETDP</strong> in the production network</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>GOMXR</td>
<td>Group or node maximum oil rate as specified with <strong>GNETDP</strong> in the production network</td>
</tr>
</tbody>
</table>
Table 2.61: Network SUMMARY output controls

**Polymer Flood Model**

The following keywords control output of data specific to the ECLIPSE 100 Polymer Flood Model:

<table>
<thead>
<tr>
<th>E100</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>six</td>
<td>E100</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>FCPR</td>
<td>GCFR</td>
<td>CCFR</td>
<td></td>
<td></td>
<td></td>
<td>Polymer Flow Rate (+ or -)</td>
</tr>
<tr>
<td>x</td>
<td>WCPR</td>
<td>GCPR</td>
<td>CCFR</td>
<td></td>
<td></td>
<td></td>
<td>Polymer Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>CCPR</td>
<td>GCPR</td>
<td>CCFR</td>
<td></td>
<td></td>
<td></td>
<td>Polymer Production Concentration</td>
</tr>
<tr>
<td>x</td>
<td>FCPC</td>
<td>GCPC</td>
<td>CCFR</td>
<td>CCPC</td>
<td>CCPC</td>
<td></td>
<td>Polymer Production Total</td>
</tr>
<tr>
<td>x</td>
<td>CCPC</td>
<td>GCPC</td>
<td>CCFR</td>
<td>CCPC</td>
<td>CCPC</td>
<td></td>
<td>Polymer Production Total</td>
</tr>
<tr>
<td>x</td>
<td>FCPT</td>
<td>GCPT</td>
<td>CCFR</td>
<td>CCPT</td>
<td>CCPT</td>
<td></td>
<td>Polymer Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>CCPT</td>
<td>GCPT</td>
<td>CCFR</td>
<td>CCPT</td>
<td>CCPT</td>
<td></td>
<td>Polymer Injection Concentration</td>
</tr>
<tr>
<td>x</td>
<td>FCIR</td>
<td>GCIR</td>
<td>CCFR</td>
<td>CCIR</td>
<td>CCIR</td>
<td></td>
<td>Polymer Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>WCIR</td>
<td>GCIR</td>
<td>CCFR</td>
<td>CCIR</td>
<td>CCIR</td>
<td></td>
<td>Polymer Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>FCIC</td>
<td>GCIC</td>
<td>CCFR</td>
<td>CCIC</td>
<td>CCIC</td>
<td></td>
<td>Polymer Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>CCIC</td>
<td>GCIC</td>
<td>CCFR</td>
<td>CCIC</td>
<td>CCIC</td>
<td></td>
<td>Polymer Injection Total</td>
</tr>
<tr>
<td>x</td>
<td>FCAD</td>
<td>GCON</td>
<td>CCFR</td>
<td>RCAD</td>
<td>BCAD</td>
<td></td>
<td>Polymer Adsorption total</td>
</tr>
<tr>
<td>x</td>
<td>RCAD</td>
<td>GCON</td>
<td>CCFR</td>
<td>BCAD</td>
<td></td>
<td></td>
<td>Polymer Adsorption concentration</td>
</tr>
<tr>
<td>E100</td>
<td>Field</td>
<td>Group</td>
<td>Well</td>
<td>Connection</td>
<td>Region</td>
<td>Block</td>
<td>Information</td>
</tr>
<tr>
<td>------</td>
<td>-------</td>
<td>-------</td>
<td>------</td>
<td>------------</td>
<td>--------</td>
<td>-------</td>
<td>-------------</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BCDCS</td>
<td>Polymer thermal degradation - total mass degraded in previous timestep</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BCDCR</td>
<td>Polymer thermal degradation - total degradation rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BCDCP</td>
<td>Polymer thermal degradation - solution degradation rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BCDCA</td>
<td>Polymer thermal degradation - adsorbed degradation rate</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BCABnnn</td>
<td>Adsorbed polymer by highest temperature band at which RRF was calculated - see keyword PLYTRRFA. The band number nnn can range from 001 to 999, but must be less than or equal to the argument of PLYTRRFA</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>CSFR</td>
<td>Salt Flow Rate (+ or -)</td>
</tr>
<tr>
<td>x</td>
<td>FSPR</td>
<td>GSPR</td>
<td>WSPR</td>
<td>CSPR</td>
<td></td>
<td></td>
<td>Salt Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>FSPT</td>
<td>GSPT</td>
<td>WSPT</td>
<td>CSPT</td>
<td></td>
<td></td>
<td>Salt Production Total</td>
</tr>
<tr>
<td>x</td>
<td>FSIR</td>
<td>GSIR</td>
<td>WSR</td>
<td>CSIR</td>
<td></td>
<td></td>
<td>Salt Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>FSIT</td>
<td>GSIT</td>
<td>WSR</td>
<td>CSIT</td>
<td></td>
<td></td>
<td>Salt Injection Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BSCN</td>
<td>Salt Cell Concentration</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Salt In Place</td>
</tr>
<tr>
<td>x</td>
<td>FSIP</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Salt In Place (as above)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>RSFT</td>
<td>Salt inter-region Flow Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BFLOW0I</td>
<td>Inter-block water flow rate in the positive I direction multiplied by the corresponding shear multiplier (PLYSHLOG and PLYSHEAR options only)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BFLOW0J</td>
<td>Inter-block water flow rate in the positive J direction multiplied by the corresponding shear multiplier (PLYSHLOG and PLYSHEAR options only)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BFLOW0K</td>
<td>Inter-block water flow rate in the positive K direction multiplied by the corresponding shear multiplier (PLYSHLOG and PLYSHEAR options only)</td>
</tr>
<tr>
<td>E100</td>
<td>Field</td>
<td>Group</td>
<td>Well</td>
<td>Connection</td>
<td>Region</td>
<td>Block</td>
<td>Information</td>
</tr>
<tr>
<td>------</td>
<td>-------</td>
<td>-------</td>
<td>------</td>
<td>------------</td>
<td>--------</td>
<td>-------</td>
<td>-------------</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BVELW0I</td>
<td>Water velocity in the positive I direction multiplied by the corresponding shear multiplier (PLYSHLOG and PLYSHEAR options only)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BVELW0J</td>
<td>Water velocity in the positive J direction multiplied by the corresponding shear multiplier (PLYSHLOG and PLYSHEAR options only)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BVELW0K</td>
<td>Water velocity in the positive K direction multiplied by the corresponding shear multiplier (PLYSHLOG and PLYSHEAR options only)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BPSHLZI</td>
<td>Viscosity multiplier due to sheared water flow in the positive I direction (PLYSHLOG and PLYSHEAR options only)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BPSHLZJ</td>
<td>Viscosity multiplier due to sheared water flow in the positive J direction (PLYSHLOG and PLYSHEAR options only)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BPSHLZK</td>
<td>Viscosity multiplier due to sheared water flow in the positive K direction (PLYSHLOG and PLYSHEAR options only)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BSRTWI</td>
<td>Water shear rate in the positive I direction following shear effects (PLYSHLOG option with keyword SHRATE)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BSRTW0J</td>
<td>Water shear rate in the positive J direction prior to shear effects (PLYSHLOG option with keyword SHRATE)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BSRTW0K</td>
<td>Water shear rate in the positive K direction prior to shear effects (PLYSHLOG option with keyword SHRATE)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BSRTWI</td>
<td>Water shear rate in the positive I direction following shear effects (PLYSHLOG option with keyword SHRATE)</td>
</tr>
</tbody>
</table>
**Table 2.62: ECLIPSE 100 Polymer flood SUMMARY output controls**

The following keywords control output of data specific to the ECLIPSE 300 Polymer Flood Model:

<table>
<thead>
<tr>
<th>E300</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>BESALTP</td>
<td>Effective salinity</td>
</tr>
<tr>
<td>x</td>
<td>BPOLY</td>
<td>Polymer concentration</td>
</tr>
<tr>
<td>x</td>
<td>BPOLYVM</td>
<td>Polymer viscosity multiplier</td>
</tr>
</tbody>
</table>

*Table 2.63: ECLIPSE 300 Polymer flood SUMMARY output controls*

**Pseudo Steady-State (ECLIPSE 100 only)**

The following keywords control output of data relevant to the calculation of a pseudo steady state solution, see keyword PSTEADY. They will automatically be output from a steady-state run if the PERFORMA keyword is present in the summary section.

<table>
<thead>
<tr>
<th>E100</th>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>PSSPR</td>
<td>Log of the pressure change per unit time, see PSTEADY</td>
</tr>
<tr>
<td>x</td>
<td>PSSSO</td>
<td>Log of the oil saturation change per unit time, see PSTEADY</td>
</tr>
<tr>
<td>x</td>
<td>PSSSW</td>
<td>Log of the water saturation change per unit time, see PSTEADY</td>
</tr>
<tr>
<td>x</td>
<td>PSSSG</td>
<td>Log of the gas saturation change per unit time, see PSTEADY</td>
</tr>
</tbody>
</table>
Reservoir coupling facility (ECLIPSE 100 only)

The following keywords control output of data relevant to the Reservoir Coupling facility:

<table>
<thead>
<tr>
<th>E100</th>
<th>Group</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>GOPRL</td>
<td>Group Oil Production Rate Target</td>
</tr>
<tr>
<td>x</td>
<td>GOIRL</td>
<td>Group Oil Injection Rate Target</td>
</tr>
<tr>
<td>x</td>
<td>GWPRL</td>
<td>Group Water Production Rate Target</td>
</tr>
<tr>
<td>x</td>
<td>GWIRL</td>
<td>Group Water Injection Rate Target</td>
</tr>
<tr>
<td>x</td>
<td>GGPRL</td>
<td>Group Gas Production Rate Target</td>
</tr>
<tr>
<td>x</td>
<td>GGIRL</td>
<td>Group Gas Injection Rate Target</td>
</tr>
<tr>
<td>x</td>
<td>GLPRL</td>
<td>Group Liquid Production Rate Target</td>
</tr>
<tr>
<td>x</td>
<td>GVPRL</td>
<td>Group reservoir Volume Production Rate Target</td>
</tr>
<tr>
<td>x</td>
<td>GVIRL</td>
<td>Group reservoir Volume Injection Rate Target</td>
</tr>
</tbody>
</table>

These keywords are intended for use with slave groups, in a slave reservoir being run on its own, when the rate limits of its slave groups are read from a Reservoir Coupling File. The keywords allow you to examine the rate targets in the file that are applied to the slave groups. These are the overall targets applied to the slave groups and represent the combination of the various limiting factors within the group tree. The reported value may therefore be less than the value reported with the same summary quantities for the master groups which report only the limits entered via the GCONPROD, GCONPRI and GCONINJE keywords.

When no rate target is being applied, the value of the limit is normally set to 1.0E20. However, the automatic axis scaling in GRAF would make the y-axis extend to this value. To avoid this, infinite rate targets are reset to zero before writing to the Summary file.

Solvent Model

The following keywords control output of data specific to the Solvent Model.

<table>
<thead>
<tr>
<th>E100</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>FNPR</td>
<td>GNPR</td>
<td>WNPR</td>
<td>CNFR</td>
<td></td>
<td></td>
<td>Solvent Flow Rate (+ or -)</td>
</tr>
<tr>
<td>x</td>
<td>FNPT</td>
<td>GNPT</td>
<td>WNPT</td>
<td>CNPT</td>
<td></td>
<td></td>
<td>Solvent Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>FNIR</td>
<td>GNIR</td>
<td>WNIR</td>
<td></td>
<td></td>
<td></td>
<td>Solvent Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>FNIT</td>
<td>GNIT</td>
<td>WNIT</td>
<td>CNIT</td>
<td></td>
<td></td>
<td>Solvent Injection Total</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BNSAT</td>
</tr>
<tr>
<td>x</td>
<td>FNIP</td>
<td></td>
<td></td>
<td></td>
<td>RNIP</td>
<td>BNIP</td>
<td>Solvent In Place</td>
</tr>
</tbody>
</table>
ECLIPSE Reference Manual

<table>
<thead>
<tr>
<th>E100</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>RNFT</td>
<td></td>
<td>Solvent inter-region Flow</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>BNKR</td>
<td></td>
<td>Solvent relative permeability</td>
</tr>
</tbody>
</table>

Table 2.66: Solvent SUMMARY output controls (ECLIPSE 100 only)

**Note:** ECLIPSE reports the field, group and well gas rates and totals for the **total** gas phase, including the solvent gas. The connection gas flow rates and totals are reported solely for the component gas phase, excluding the solvent gas.

### Surfactant Model

The following keywords control output of data specific to the Surfactant Model.

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FTPRSUR</td>
<td>GTPRSUR</td>
<td>WTPRSUR</td>
<td>CTFRSUR</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FTPTSUR</td>
<td>GTPSUR</td>
<td>WTPTSUR</td>
<td>CTPTSUR</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FTIRSUR</td>
<td>GTIRSUR</td>
<td>WTIRSUR</td>
<td>CTITSUR</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FTITSUR</td>
<td>GTITSUR</td>
<td>WTITSUR</td>
<td>CTITSUR</td>
</tr>
</tbody>
</table>

Table 2.67: Surfactant SUMMARY field, group, well and connection output controls

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Field</th>
<th>Region</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BTCNFSUR</td>
<td></td>
<td>Concentration</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BSURF</td>
<td></td>
<td>Concentration in solution</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FTIPTSUR</td>
<td>RTIPTSUR</td>
<td>BPTIPTSUR</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>RTFTTSUR</td>
<td></td>
<td>Inter-region Flow Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>FTADSUR</td>
<td>RTADSUR</td>
<td>Adsorption total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BTADSUR</td>
<td></td>
<td>Adsorption</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BTCASUR</td>
<td></td>
<td>Log (Capillary Number)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BSURFCNM</td>
<td></td>
<td>Log (Capillary Number)</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BTSTSUR</td>
<td></td>
<td>Surface tension</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BSURFST</td>
<td></td>
<td>Surface tension</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BEWV_SUR</td>
<td></td>
<td>Effective water viscosity due to surfactant concentration</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BSURFKR</td>
<td></td>
<td>Relative permeability interpolation factor</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BSURFCP</td>
<td></td>
<td>Capillary pressure multiplier</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>BESALTS</td>
<td></td>
<td>Effective salinity</td>
</tr>
</tbody>
</table>

Table 2.68: Surfactant SUMMARY field, region and block output controls
Alkaline model

The following keywords control output of data specific to the Alkaline Model.

<table>
<thead>
<tr>
<th>E100</th>
<th>Field</th>
<th>Group</th>
<th>Well</th>
<th>Connection</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>FTPALK</td>
<td>GTPALK</td>
<td>WTPALK</td>
<td>CTFRALK</td>
<td></td>
<td>Flow Rate (+ or -)</td>
</tr>
<tr>
<td>x</td>
<td>FTPTALK</td>
<td>GTPTALK</td>
<td>WPTALK</td>
<td>CGPTALK</td>
<td></td>
<td>Production Rate</td>
</tr>
<tr>
<td>x</td>
<td>FTIRALK</td>
<td>GTIRALK</td>
<td>WIRALK</td>
<td>CTIRALK</td>
<td></td>
<td>Injection Rate</td>
</tr>
<tr>
<td>x</td>
<td>FTITALK</td>
<td>GTITALK</td>
<td>WITALK</td>
<td>CTITALK</td>
<td></td>
<td>Injection Total</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BTCNFALK</td>
<td></td>
<td>Concentration</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BTADSALK</td>
<td></td>
<td>Adsorption</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BTSTMALK</td>
<td></td>
<td>Surface tension multiplier</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BTSADALK</td>
<td></td>
<td>Surfactant adsorption multiplier</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td>BTPADALK</td>
<td></td>
<td>Polymer adsorption multiplier</td>
</tr>
</tbody>
</table>

*Table 2.69: Alkaline SUMMARY output controls (ECLIPSE 100 only)*

**Note:** When requesting output of alkaline or surfactant (or any other active tracer) properties on an LGR, the corresponding SUMMARY mnemonics may exceed 8 characters in length, for example LBTCNFALK or LBTIPTSUR. In these special cases ECLIPSE will recognize the truncated forms of the mnemonics, that is LBTCNFFSU and LBTIPTAL.

<table>
<thead>
<tr>
<th>E300</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>BALSURF</td>
<td>Alkaline concentration</td>
</tr>
<tr>
<td>x</td>
<td>BALSTML</td>
<td>Surface tension multiplier</td>
</tr>
</tbody>
</table>

*Table 2.70: Alkaline SUMMARY output block controls (ECLIPSE 300 only)*

Wellbore friction model

The following items are relevant to the Wellbore Friction Model:

<table>
<thead>
<tr>
<th>E100</th>
<th>Connection</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>COFRU</td>
<td>Sum of connection oil flow rates upstream of, and including, this connection</td>
</tr>
<tr>
<td>x</td>
<td>CWFRU</td>
<td>Sum of connection water flow rates upstream of, and including, this connection</td>
</tr>
<tr>
<td>x</td>
<td>CGFRU</td>
<td>Sum of connection gas flow rates upstream of, and including, this connection</td>
</tr>
<tr>
<td>x</td>
<td>LCOFRU</td>
<td>As COFRU but for local grids</td>
</tr>
<tr>
<td>x</td>
<td>LCWFRU</td>
<td>As CWFRU but for local grids</td>
</tr>
<tr>
<td>x</td>
<td>LGFRU</td>
<td>As CGFRU but for local grids</td>
</tr>
</tbody>
</table>

*Table 2.71: Wellbore friction SUMMARY output controls (ECLIPSE 100)*

Use the local grid keywords if the well is situated in a local grid.
Conductive Faults

The following items are relevant to the Conductive Fault model:

<table>
<thead>
<tr>
<th>E300</th>
<th>Conductive Fault</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>CFOWC</td>
<td>Oil-water contact depth in the fault</td>
</tr>
<tr>
<td>x</td>
<td>CFGOC</td>
<td>Gas-oil contact depth in the fault</td>
</tr>
<tr>
<td>x</td>
<td>CPFRES</td>
<td>Pressure in the fault</td>
</tr>
<tr>
<td>x</td>
<td>CFOSAT</td>
<td>Oil saturation in the fault</td>
</tr>
<tr>
<td>x</td>
<td>CFGSAT</td>
<td>Gas saturation in the fault</td>
</tr>
<tr>
<td>x</td>
<td>CFWSAT</td>
<td>Water saturation in the fault</td>
</tr>
</tbody>
</table>

Table 2.72: Conductive Fault SUMMAR Y output controls (ECLIPSE 300)

User defined quantities

The user defined summary output options are:

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>User defined quantity</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>CUxxxxxxx</td>
<td>User defined connection quantity</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FUxxxxxxx</td>
<td>User defined field quantity</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>GUxxxxxxx</td>
<td>User defined group quantity</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>RUxxxx</td>
<td>User defined region quantity</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SUxxxxxxx</td>
<td>User defined segment quantity</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>WUxxxxxxx</td>
<td>User defined well quantity</td>
</tr>
</tbody>
</table>

Table 2.73: User defined quantities SUMMARY output controls

User defined summary quantities are constructed using the UDQ keyword in the SCHEDULE section. The facility allows the user to define quantities as either constants, or composed of ECLIPSE summary quantities in conjunction with a number of mathematical functions.

ODD3P three-phase relative permeability and capillary pressure hysteresis model

The following items are specific to the ODD3P three-phase relative permeability and capillary pressure hysteresis model:

<table>
<thead>
<tr>
<th>E300</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>BSONRM</td>
<td>Normalized oil saturation</td>
</tr>
<tr>
<td>x</td>
<td>BSGNRM</td>
<td>Normalized gas saturation</td>
</tr>
<tr>
<td>x</td>
<td>BSWNRM</td>
<td>Normalized water saturation</td>
</tr>
<tr>
<td>x</td>
<td>BSDROW</td>
<td>Oil phase saturation direction indicator for oil-water system</td>
</tr>
<tr>
<td>x</td>
<td>BSRWO</td>
<td>Water phase saturation direction indicator for water-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BSDROG</td>
<td>Oil phase saturation direction indicator for oil-gas system</td>
</tr>
<tr>
<td>E300</td>
<td>Block</td>
<td>Information</td>
</tr>
<tr>
<td>------</td>
<td>------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>x</td>
<td>BSDRGO</td>
<td>Gas phase saturation direction indicator for gas-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BSDRGW</td>
<td>Gas phase saturation direction indicator for gas-water system</td>
</tr>
<tr>
<td>x</td>
<td>BSDRWG</td>
<td>Water phase saturation direction indicator for water-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BSWNT</td>
<td>Normalized oil turning point saturation for oil-water system.</td>
</tr>
<tr>
<td>x</td>
<td>BSWONT</td>
<td>Normalized water turning point saturation for water-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BSOGNT</td>
<td>Normalized oil turning point saturation for oil-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BSGONT</td>
<td>Normalized gas turning point saturation for gas-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BSGWNT</td>
<td>Normalized gas turning point saturation for gas-water system</td>
</tr>
<tr>
<td>x</td>
<td>BSOWNH</td>
<td>Normalized oil hysteresis saturation for oil-water system</td>
</tr>
<tr>
<td>x</td>
<td>BSWONH</td>
<td>Normalized water hysteresis saturation for water-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BSOGNH</td>
<td>Normalized oil hysteresis saturation for oil-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BSGONH</td>
<td>Normalized gas hysteresis saturation for gas-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BSGWNH</td>
<td>Normalized gas hysteresis saturation for gas-water system</td>
</tr>
<tr>
<td>x</td>
<td>BSWGNH</td>
<td>Normalized water hysteresis saturation for gas-water system</td>
</tr>
<tr>
<td>x</td>
<td>BSOWNE</td>
<td>Normalized oil equivalent opposite direction saturation for oil-water system</td>
</tr>
<tr>
<td>x</td>
<td>BSWONE</td>
<td>Normalized water equivalent opposite direction saturation for water-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BSOGNE</td>
<td>Normalized oil equivalent opposite direction saturation for oil-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BSGONE</td>
<td>Normalized gas equivalent opposite direction saturation for gas-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BSGWNE</td>
<td>Normalized gas equivalent opposite direction saturation for gas-water system</td>
</tr>
<tr>
<td>x</td>
<td>BSWGNE</td>
<td>Normalized water equivalent opposite direction saturation for water-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BSORWB</td>
<td>Residual oil saturation used to calculate normalized saturations in oil-water system</td>
</tr>
<tr>
<td>x</td>
<td>BSWROB</td>
<td>Residual water saturation used to calculate normalized saturations in water-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BSORGB</td>
<td>Residual oil saturation used to calculate normalized saturations in oil-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BSGROB</td>
<td>Residual gas saturation used to calculate normalized saturations in gas-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BSGRWB</td>
<td>Residual gas saturation used to calculate normalized saturations in gas-water system</td>
</tr>
<tr>
<td>x</td>
<td>BSWRGB</td>
<td>Residual water saturation used to calculate normalized saturations in water-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BSORWP</td>
<td>Process dependent residual oil saturation in oil-water system</td>
</tr>
<tr>
<td>x</td>
<td>BSWROP</td>
<td>Process dependent residual water saturation in water-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BSORGP</td>
<td>Process dependent residual oil saturation in oil-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BSGROP</td>
<td>Process dependent residual gas saturation in gas-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BSGRWP</td>
<td>Process dependent residual gas saturation in gas-water system</td>
</tr>
<tr>
<td>x</td>
<td>BSWRGP</td>
<td>Process dependent residual water saturation in water-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BKROWR</td>
<td>Representative oil-water relative permeability</td>
</tr>
<tr>
<td>x</td>
<td>BKRWOR</td>
<td>Representative water-oil relative permeability</td>
</tr>
<tr>
<td>x</td>
<td>BKROGR</td>
<td>Representative oil-gas relative permeability</td>
</tr>
<tr>
<td>E300</td>
<td>Block</td>
<td>Information</td>
</tr>
<tr>
<td>------</td>
<td>--------</td>
<td>-----------------------------------------------------------------</td>
</tr>
<tr>
<td>x</td>
<td>BKRGOR</td>
<td>Representative gas-oil relative permeability</td>
</tr>
<tr>
<td>x</td>
<td>BKRGWR</td>
<td>Representative gas-water relative permeability</td>
</tr>
<tr>
<td>x</td>
<td>BKRWGR</td>
<td>Representative water-gas relative permeability</td>
</tr>
<tr>
<td>x</td>
<td>BKROWH</td>
<td>Turning point relative permeability to oil for oil-water system</td>
</tr>
<tr>
<td>x</td>
<td>BKRWOH</td>
<td>Turning point relative permeability to water for water-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BKROGH</td>
<td>Turning point relative permeability to oil for oil-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BKRGOH</td>
<td>Turning point relative permeability to gas for gas-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BKRGWH</td>
<td>Turning point relative permeability to gas for gas-water system</td>
</tr>
<tr>
<td>x</td>
<td>BKRWGH</td>
<td>Turning point relative permeability to water for water-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BKRWE</td>
<td>Equivalent relative permeability to oil for oil-water system</td>
</tr>
<tr>
<td>x</td>
<td>BKRWOE</td>
<td>Equivalent relative permeability to water for water-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BKROGE</td>
<td>Equivalent relative permeability to oil for oil-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BKRGOE</td>
<td>Equivalent relative permeability to gas for gas-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BKRGWE</td>
<td>Equivalent relative permeability to gas for gas-water system</td>
</tr>
<tr>
<td>x</td>
<td>BKRWGE</td>
<td>Equivalent relative permeability to water for water-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BKROWT</td>
<td>Opposite saturation direction turning point relative permeability to oil for oil-water system</td>
</tr>
<tr>
<td>x</td>
<td>BKRWOT</td>
<td>Opposite saturation direction turning point relative permeability to water for water-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BKROGT</td>
<td>Opposite saturation direction turning point relative permeability to oil for oil-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BKRGOT</td>
<td>Opposite saturation direction turning point relative permeability to gas for gas-oil system</td>
</tr>
<tr>
<td>x</td>
<td>BKRGWT</td>
<td>Opposite saturation direction turning point relative permeability to gas for gas-water system</td>
</tr>
<tr>
<td>x</td>
<td>BKRWGT</td>
<td>Opposite saturation direction turning point relative permeability to water for water-gas system</td>
</tr>
<tr>
<td>x</td>
<td>BIFTOW</td>
<td>Oil-water interfacial tension</td>
</tr>
<tr>
<td>x</td>
<td>BIFTWO</td>
<td>Water-oil interfacial tension (alias for BIFTOW)</td>
</tr>
<tr>
<td>x</td>
<td>BIFTOG</td>
<td>Oil-gas interfacial tension (alias for BIFTGO)</td>
</tr>
<tr>
<td>x</td>
<td>BIFTGO</td>
<td>Gas-oil interfacial tension</td>
</tr>
<tr>
<td>x</td>
<td>BIFTGW</td>
<td>Gas-water interfacial tension</td>
</tr>
<tr>
<td>x</td>
<td>BIFTWG</td>
<td>Water-gas interfacial tension (alias for BIFTWG)</td>
</tr>
<tr>
<td>x</td>
<td>BPCOWR</td>
<td>Representative oil-water capillary pressure</td>
</tr>
<tr>
<td>x</td>
<td>BPCWOR</td>
<td>Representative water-oil capillary pressure</td>
</tr>
<tr>
<td>x</td>
<td>BPCGOR</td>
<td>Representative oil-gas capillary pressure</td>
</tr>
<tr>
<td>x</td>
<td>BPCGOR</td>
<td>Representative gas-oil capillary pressure</td>
</tr>
<tr>
<td>x</td>
<td>BPCGWR</td>
<td>Representative gas-water capillary pressure</td>
</tr>
</tbody>
</table>
**Asphaltene model**

The following outputs are specific to the Asphaltene model activated using the `ASPHALTE` keyword in the `RUNSPEC` section. Please refer to "Asphaltene Option" in the *ECLIPSE Technical Description* for more details. See also the notes for this table:

1. See “Format of SUMMARY keywords with a component index”.

<table>
<thead>
<tr>
<th>E300</th>
<th>Block</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>BPCWGR</td>
<td>Representative water-gas capillary pressure</td>
</tr>
</tbody>
</table>

*Table 2.74: ODD3P SUMMARY output controls (ECLIPSE 300 only)*

**Simulator performance keywords**

The keywords are:

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>ELAPSED</td>
<td>Elapsed time in seconds.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>HLINEARS</td>
<td>Number of linear iterations for each gradient calculation (Gradient Option).</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>HSUMLINS</td>
<td>Total number of linear iterations for each gradient pressure (Gradient Option).</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>MAXCFL</td>
<td>Maximum explicit CFL value for each timestep.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>MAXDE</td>
<td>Maximum change in energy density for each timestep.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MAXDPR</td>
<td>Maximum change in pressure for each timestep.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MAXDSG</td>
<td>Maximum change in gas saturation for each timestep.</td>
</tr>
</tbody>
</table>

*Table 2.75: Asphaltene SUMMARY output controls (ECLIPSE 300)*
<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>MAXDSO</td>
<td>Maximum change in oil saturation for each timestep.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MAXDSW</td>
<td>Maximum change in water saturation for each timestep.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>MAXDT</td>
<td>Maximum change in temperature for each timestep.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MEMORYTS</td>
<td>Maximum current memory usage across processors as reported by the operating system.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MLINEARP</td>
<td>Number of pressure iterations for each timestep (CPR or JALS solver only).</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MLINEARS</td>
<td>Number linear iterations for each timestep.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MSUMLINP</td>
<td>Total number of pressure iterations since the start of the run (CPR or JALS solver only).</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MSUMBUG</td>
<td>Total number of ‘bug’ messages since the start of the run.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MSUMCOMM</td>
<td>Total number of ‘comment’ messages since the start of the run.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MSUMERR</td>
<td>Total number of ‘error’ messages since the start of the run.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MSUMLINS</td>
<td>Total number of linear iterations since the start of the run.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MSUMMESS</td>
<td>Total number of ‘message’ messages since the start of the run.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MSUMNEWT</td>
<td>Total number of Newton iterations since the start of the run.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MSUMPROB</td>
<td>Total number of ‘problem’ messages since the start of the run.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>MSUMWARN</td>
<td>Total number of ‘warning’ messages since the start of the run.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>NAIMFRAC</td>
<td>Fraction of implicit cells in AIM run.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>NBAKFL</td>
<td>Number of calls made to scalar fallback routines in the flash calculation for each timestep.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>NBYTOT</td>
<td>Peak usage of dynamically allocated memory as reported the simulator internal memory allocator. Note that some memory allocations are performed by other mechanisms and are not accounted for by this summary quantity. In parallel, this is the maximum across processors.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>NCPRLINS</td>
<td>Average number of pressure iterations per linear iteration for each timestep (CPR or JALS solver only).</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>NEWTFL</td>
<td>Cumulative average Newton iterations per cell in two phase flash calculations.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>NEWTON</td>
<td>Number of Newton iterations used for each timestep</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>NLINEARP</td>
<td>Average number of pressure iterations per Newton iteration for each timestep (CPR or JALS solver only).</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>NLINEARS</td>
<td>Average number of linear iterations per Newton iteration for each timestep.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>NLINSMIN</td>
<td>Actual minimum number of linear iterations in the Newton iterations within each timestep.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>NLINSMAX</td>
<td>Actual maximum number of linear iterations in the Newton iterations within each timestep.</td>
</tr>
<tr>
<td>E100</td>
<td>E300</td>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>------</td>
<td>------</td>
<td>---------</td>
<td>-------------</td>
</tr>
</tbody>
</table>
| x    | x    | NMESSAGE| Requests a set of message data vectors for the run. The set of keywords implied by NMESSAGE is:  
  MSUMBUG, MSUMCOMM, MSUMERR, MSUMMESS, MSUMPROB and MSUMWARN |
| x    |      | NNUMFL  | Total number of two phase flash calculations performed. |
| x    |      | NNUMST  | Total number of stability tests performed. |
| x    |      | NLRESSUM| Sum of the non-linear residual. |
| x    |      | NLRESMAX| Maximum element of the non-linear residual. |
| x    |      | NTS     | Number of timesteps taken. |
| x    |      | NTSPCL  | Number of timesteps in which the pressure solution variable converged last.  
The tolerance on the pressure solution change can be modified using item 1 of the CVCRIT keyword. |
| x    |      | NTSMCL  | Number of timesteps in which the molar density solution variables converged last.  
The tolerance on the pressure solution change can be modified using item 7 of the CVCRIT keyword. |
| x    |      | NTSECL  | Number of timesteps in which the energy density solution variable converged last.  
The tolerance on the pressure solution change can be modified using item 8 of the CVCRIT keyword (THERMAL option) |
| x    |      | PERFORMA| Requests a set of performance data vectors for the run. The set of keywords implied by PERFORMA is:  
  TCPU, ELAPSED, NEWTON, NLINEARS, NLINSMIN, NLINSMAX, MLINEARS, MSUMLINS, MSUMNEWT, TIMESTEP, TCPUTS, TCPUDAY, STEPTYPE, and TELAPLIN. If a steady-state run, set with keyword PSTEADY, then the PSS® keywords are also output.  
  If the CPR solver has been selected MLINEARP, MSUMLINP, NCPRLINS, NLINEARP are additionally output. |
| x    |      | PERFORMA| Requests a set of performance data vectors for the run. The set of keywords implied by PERFORMA is:  
  TCPU, ELAPSED, NEWTON, NLINEARS, NLINSMIN, NLINSMAX, MLINEARS, MSUMLINS, MSUMNEWT, TIMESTEP, NEWTFL, NNUMFL, NNUMST, NBACKFL, TCPUTS, TCPUDAY.  
  If the AIM Solution option has been selected NAIMFRAC is additionally output.  
  If the CPR solver or the JALS solver has been selected MLINEARP, MSUMLINP, NCPRLINS, NLINEARP are additionally output. |
### Data File Overview

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
</table>
| x    |      | STEPTYPE  | Outputs an integer representing the reason for selecting the timestep length (see "Convergence Reports" in the ECLIPSE Technical Description): 1=INIT 2=TRNC 3=MNF 4=MAXF 5=MINS 6=MAXS 7=REPT 8=HALF 9=CHOP 10=SATM 11=PCHP 12=DIFF 13=LGR C 14=SURF 15=NETW 16=THRP 17=EMTH 18=MAXP 19=WCYC 20=MAST 21=SLVR 22=SLVC 23=MAXW 24=EFF+ 25=EFF- 26=NLTR 27=EFFT 28=DLYA 29=ACTN 30=RAIN  
For initial time, the STEPTYPE will remain blank.  
When a neat tabulated output of the summary file data is printed in the Print file at the end of a run (see RUNSUM) or in a separate RSM file (see SEPARATE, EXCEL and LOTUS), then the output is rendered as mnemonics. |
| x    |      | TCPU      | Current CPU usage in seconds.  
This does not take into account the time taken by inter-process communications, whereas ELAPSED does. Consequently, in parallel, ELAPSED is the most relevant time measurement. |
| x    |      | TCPUDAY   | CPU time per day (or hour in lab units). |
| x    |      | TCPUH     | Total CPU time for each gradient calculation (Gradient Option). |
| x    |      | TCUHT     | Total CPU time for all gradient calculations (Gradient Option). |
| x    |      | TCPUZH    | Total CPU time used in SCHEDULE section. |
| x    |      | TCPUTSH   | CPU time per timestep for each gradient calculation (Gradient Option). |
| x    |      | TCPUTSHT  | CPU time per timestep for all gradient calculations (Gradient Option). |
| x    |      | TCPUTR    | Total CPU time used in tracer calculations. |
| x    |      | TCPUTRSV  | Total CPU time used in tracer calculations by the implicit solver. |
| x    |      | TCPULGTR  | Total CPU time used in LAGTRAC streamline tracing. |
| x    |      | TCPULGSV  | Total CPU time used in the LAGTRAC tracer solver. |
| x    |      | TELAPDAY  | Elapsed time per day (or hour in lab units). |
| x    |      | TELAPLIN  | Elapsed time per linear iteration in seconds. |
| x    |      | TELAPTS   | Elapsed time per timestep in seconds. |
| x    |      | TIMESTEP  | Timestep length. |
| x    |      | TUNE      | Requests the following set of vectors for the run: NLRESSUM, NLRESMAX, NTS, NTSPCL, NTSMCL and NTSECL in a THERMAL simulation. |
| x    |      | WNEWTON   | Number of well Newton iterations taken in the last global Newton iteration.  
A negative value indicates that the well failed to converge. |
| x    |      | ZIPEFF    | Predicted efficiency of the timestep (mainly for developer use). |
Table 2.76: Simulator performance keywords

Note: When requesting output of total number of messages with the MSUMBUG, MSUMCOMM, MSUMERR, MSUMMESS, MSUMPROB, MSUMWARN or NMESSAGE for all message mnemonics, the count(s) reported by these mnemonics may be truncated by one count if the last type of message reported causes the run to stop.

Other keywords

The other SUMMARY keywords are listed below.

ALL Requests a basic set of field, group and well keywords for all groups and wells present in the problem. The set of keywords implied by ALL is:

- FOPR  GOPR  WOPR  FOPT  GOPT  WOPT
- FOIR  GOIR  WOIR  FOIT  GOIT  WOIT  These keywords are only valid in ECLIPSE 100.
- FWPR  GWPR  WWPR  FWPT  GWPT  WWPT
- FWIR  GWIR  WWIR  FWIT  GWIT  WWIT
- FGPR  GGPR  WGRP  FGPT  GGPT  WGPFT
- FGIR  GGIR  WGIR  FGIT  GGIT  WGIT
- FVPR  GVPR  WVPR  FVPT  GVPT  WVPT
- FVIR  GVIR  WVIR  FVIT  GVIT  WVIT
- FWCT  GWCT  WWCT  FGOR  GGOR  WGOR
- FWGR  GWGR  WWGR
- WBHP  WTHP  WPI
- FOIP  FOIPL  FOIPG
- FWIP
- FGIP  FG IPL  FGIPG  FPR
- FAQR  FAQRG  AAQR  AAQRG
- FAQT  FAQTG  AAQT  AAQTG

Table 2.76: Simulator performance keywords

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>DATE</td>
<td>Requests the date as three summary quantities DAY, MONTH, YEAR. In addition, the date is printed in place of the time in the RUNSUM output</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>EXCEL</td>
<td>Requests RUNSUM output to be in Microsoft Excel format.</td>
</tr>
</tbody>
</table>
Table 2.77: Other SUMMARY keywords

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
<td>GMWSET</td>
<td>Requests the set of keywords that counts the number of production and injection wells in various states and under various control modes. The set also includes counts of drilling queue and workover events in a timestep and the cumulative totals of such events. The wells included in the counts are limited to those that can be hierarchically traced to a named group.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>FMWSET</td>
<td>Requests the set of mnemonics that counts the number of all production and injection wells in various states and under various control modes. The set also includes counts of drilling queue and workover events in a timestep and the cumulative totals of such events.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>LOTUS</td>
<td>Requests RUNSUM output to be in Lotus 1-2-3 format.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>MONITOR</td>
<td>Outputs information required by GRAF for run-time monitoring.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>NOMONITO</td>
<td>Disables run-time monitoring.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>NARROW</td>
<td>Increases the number of columns per page in the RUNSUM output by restoring the pre-97A (ECLIPSE 100) or pre-2000A (ECLIPSE 300) format with narrower columns.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>RPTONLY</td>
<td>Requests that the summary data is only produced at report times. The default if RPTONLY is not present, is to write the summary information at every time.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>RPTONLY</td>
<td>Requests summary data is written every timestep. This is the default situation and the keyword need only be used to turn off a previously entered RPTONLY instruction.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>RPTSMRY</td>
<td>Requests printed output of keywords in the SUMMARY section</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>RUNSUM</td>
<td>Requests a neat tabulated output of the summary file data at the end of the normal ECLIPSE 100 printed output.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>RUNSUM</td>
<td>Requests a neat tabulated output of the summary file data at the end of the run, this output goes to a separate RSM file.</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>SEPARATE</td>
<td>Requests RUNSUM output to go to a separate RSM file.</td>
</tr>
<tr>
<td>x</td>
<td>x</td>
<td>SUMTHIN</td>
<td>Reduces the amount of summary data recorded. Data are only output once in each time period defined by this keyword, as well as every report timestep.</td>
</tr>
</tbody>
</table>

**Deprecated keywords**

The following SUMMARY keywords have been deprecated due to naming convention irregularities. The following replacements should be used:

<table>
<thead>
<tr>
<th>E300</th>
<th>Deprecated Keyword</th>
<th>Replacement Keyword</th>
<th>Deprecation release</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>CCFR</td>
<td>CKFR</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>CCFT</td>
<td>CKFT</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>RCFR</td>
<td>RKFR</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>RCFR+</td>
<td>RKFR+</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>RCFR−</td>
<td>RKFR−</td>
<td>2011.1</td>
</tr>
</tbody>
</table>
Table 2.78: Deprecated SUMMARY keywords

<table>
<thead>
<tr>
<th>E300</th>
<th>Deprecated Keyword</th>
<th>Replacement Keyword</th>
<th>Deprecation release</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>RCFT</td>
<td>RKFT</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>RCFT+</td>
<td>RKFT+</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>RCFT−</td>
<td>RKFT−</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>BSWL</td>
<td>BPSWL</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>BSWCR</td>
<td>BPSWCR</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>BSWU</td>
<td>BPSWU</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>BSGL</td>
<td>BPSGL</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>BSGCR</td>
<td>BPSGCR</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>BSGU</td>
<td>BPSGU</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>BSOWC</td>
<td>BPSOWCR</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>BSOGC</td>
<td>BPSOGCR</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>BCFRI</td>
<td>BKFRI</td>
<td>2012.1</td>
</tr>
<tr>
<td>x</td>
<td>BCFRJ</td>
<td>BKFRJ</td>
<td>2012.1</td>
</tr>
<tr>
<td>x</td>
<td>BCFRK</td>
<td>BKFRK</td>
<td>2012.1</td>
</tr>
<tr>
<td>x</td>
<td>SCFRn</td>
<td>SCWGFR</td>
<td>2012.1</td>
</tr>
<tr>
<td>x</td>
<td>BPORVT</td>
<td>BRPV</td>
<td>2014.1</td>
</tr>
<tr>
<td>x</td>
<td>LBPORVT</td>
<td>LBRPV</td>
<td>2014.1</td>
</tr>
</tbody>
</table>

Table 2.79: Deleted SUMMARY keywords

**Deleted keywords**

The following SUMMARY keywords have been deleted. The following replacements must be used:

<table>
<thead>
<tr>
<th>E300</th>
<th>Deleted Keyword</th>
<th>Replacement Keyword</th>
<th>Deprecation release</th>
<th>Deletion Release</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>BASPFL</td>
<td>BAREAC</td>
<td>2011.1</td>
<td>2011.1</td>
</tr>
<tr>
<td>x</td>
<td>WNPR</td>
<td>WNLPR</td>
<td>2008.1</td>
<td>2012.1</td>
</tr>
<tr>
<td>x</td>
<td>WNPTR</td>
<td>WNLPT</td>
<td>2008.1</td>
<td>2012.1</td>
</tr>
<tr>
<td>x</td>
<td>GNPR</td>
<td>GNLPR</td>
<td>2008.1</td>
<td>2012.1</td>
</tr>
<tr>
<td>x</td>
<td>GNPT</td>
<td>GNLPT</td>
<td>2008.1</td>
<td>2012.1</td>
</tr>
<tr>
<td>x</td>
<td>FNPR</td>
<td>FNLPR</td>
<td>2008.1</td>
<td>2012.1</td>
</tr>
<tr>
<td>x</td>
<td>FNPT</td>
<td>FNLPT</td>
<td>2008.1</td>
<td>2012.1</td>
</tr>
<tr>
<td>x</td>
<td>RNPRI</td>
<td>RNLIP</td>
<td>2008.1</td>
<td>2012.1</td>
</tr>
<tr>
<td>x</td>
<td>RNPR</td>
<td>RNLPR</td>
<td>2008.1</td>
<td>2012.1</td>
</tr>
<tr>
<td>x</td>
<td>RNPT</td>
<td>RNLPT</td>
<td>2008.1</td>
<td>2012.1</td>
</tr>
</tbody>
</table>
SCHEDULE section

The SCHEDULE section specifies the operations to be simulated (production and injection controls and constraints) and the times at which output reports are required. Vertical flow performance curves and simulator tuning parameters may also be specified in the SCHEDULE section.

All keywords in this section are optional, except for those necessary to define the status of the wells, and the END keyword, which should mark the end of the scheduling data.

To define a well and its connection properties and controls, the following keywords should be used:

1. WELSPECS or WELSPECL (to introduce the well).

   Having created a well specification you can now set its completion and control data.

2. COMPDAT or COMPDATL (to specify its completion data)

3. You can specify well controls using the following keywords:
   WCONPROD (production controls, if the well is a producer) or
   WCONINJE (injection controls, if the well is an injector) or
   WCONHIST (measured flows and pressures, if it is a history matching producer) or
   WCONINJH (measured flows and pressures, if it is a history matching injector) or
   WCONINJP (control of a pattern flood injector)

4. In ECLIPSE 300, separators should be defined with SEPCOND and allocated to wells with WSEPCOND. Injection gas composition should also be defined with WINJGAS or GINJGAS. Injection oil composition should be defined with the WINJOIL keyword.

The keywords WELSPECL and COMPDATL should be used in local grid refinement runs when the well is situated in a local grid. There are also some alternatives to the above keywords that are available only in ECLIPSE 300: WELSPEC, WELLCOMP, WELLPROD and WELLINJE. These original keywords are retained for back compatibility, but you are encouraged to use the keywords that are compatible with both simulators.

Any other keywords that refer to a particular well must be positioned after the well and its connections have been defined.

Wells can be introduced at any time in the simulation, but once a well has been introduced using the keyword WELSPECS or WELSPECL its connection properties and operating status must be defined as shown above. Data concerning the well can be changed later in the simulation by repeating the appropriate keyword(s).

Certain keywords (for example GCONPROD, GCONINJE, GECON) refer to groups. A new group is automatically introduced immediately its name appears in either keyword WELSPECS or WELSPECL its connection properties and operating status must be defined as shown above. Data concerning the well can be changed later in the simulation by repeating the appropriate keyword(s).

Well and group names, and other character strings in the scheduling data, may be enclosed in quotes (""), but generally this is not essential. Such quotes are only usually required if a name contains embedded blanks, starts with a number or contains non-alphanumeric characters. Quotes are also required for name roots and well list names (see below), which contain an asterisk (*)

In most of the well scheduling keywords a well name root, ending with an asterisk (*), can be used to refer to several wells in one record. The data is applied to all currently defined wells whose names begin with the sequence of characters preceding the asterisk in the name root. For example the root ‘PR* ’ applies to all
wells currently entered with names beginning with PR, including the name ‘PR’ itself. The root ‘*’ will refer to all wells currently entered. It is important to note that the data does not apply to any wells introduced later in the run. Therefore if you are going to use well name roots, it may be advisable to define all the wells at the beginning of the run with the keywords WELSPECS, COMPDAT and WCONPROD or WCONINJE or WCONHIST or WCONINJH or WCONINJP and shut the wells that are not needed until later. Group name roots can similarly be used to refer to multiple groups. As with well name roots, group name roots only refer to groups that are currently defined. Well and group name roots must not have any characters following the asterisk.

An alternative method of referring to several wells in one record is to use a well list name. These begin with an asterisk, followed by at least one letter. A well list comprises a set of well names; there is no limit to the number or length of the well lists, and they may be changed at any time in the simulation. The keywords WLIST and WLISTDYN (ECLIPSE 300 only) provide a set of facilities for constructing and managing well lists.

All keywords must start in column 1 of the data file. The keywords may be up to 8 characters long, and all 8 characters (including blanks) are significant. Most of the keywords have a similar syntax: the keyword is followed by any number of records, each containing a number of data items, and each terminated by a slash character (/). The records can be terminated early using the slash character (/); the remaining items take their default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted. The set of records is ended by a blank record, containing only a slash. Some of the data items are character strings which select a particular option from a set of available options. In general, these may be truncated to the first one or two characters as necessary to avoid ambiguity. However, you are recommended to enter the complete character string as some sets of options may be extended in future releases and the number of characters necessary to avoid ambiguity may increase.

The order in which keywords are specified in the SCHEDULE section dictates the order in which the specified operations are simulated. A TIME, TSTEP or DATES keyword advances the simulation to a specified time. A subsequent well control keyword alters the well status from the current time. A further TIME, TSTEP or DATES keyword advances the simulation through another time period using the new well status. For example, if a WELTARG keyword is used to redefine a well production rate at a certain time, a subsequent TIME, TSTEP or DATES keyword advances the simulation to the next report time with the well operating at its new rate.

A list of keywords available in the SCHEDULE section is given below, together with a brief description of their function. The list is grouped into functional categories. The list does not include old-style ECLIPSE 300 keywords which have been superseded by ECLIPSE 100 compatible keywords, although these keywords are generally retained for back compatibility. The list also does not include keywords specific to the special extensions (for example, local grid refinement); these are listed in the appropriate chapter of the ECLIPSE Technical Description. A more detailed description of all the keyword data may be found by looking up the keyword in this manual.

### Define wells, groups and well connections

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required</td>
<td>WELSPECS</td>
<td>Introduces new wells and specifies some of their general data, including the group to which they belong. A well must be introduced with this keyword before it can be referenced in any other keyword.</td>
</tr>
</tbody>
</table>
Restriction | Keyword | Description
--- | --- | ---
 | GRUPTREE | Defines a multi-level tree structure for groups. This keyword is only required if there is more than one level of groups in the field. Well-groups (that is, groups containing wells) are introduced by naming them in WELSPECS. In a multi-level group structure there are node-groups, that is groups that have other groups as their immediate subordinates instead of wells. Node groups are introduced by naming them in GRUPTREE.

**Note:** If the GRUPTREE keyword is not used, all groups are assumed to be immediately subordinate to the FIELD, which is the name reserved for the top-most node in the structure. The FIELD cannot have wells of its own, and should not appear in WELSPECS. Apart from the FIELD, a group must have been introduced in WELSPECS or GRUPTREE before it can be referenced in any other keyword.

---

**Define well connections and set connection properties**

**Common keywords**

<table>
<thead>
<tr>
<th>Restrictions</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>COMPDAT</td>
<td>Defines connections between wells and grid blocks. This keyword is compulsory. All wells must have at least one connection defined. Connections are identified by their well name together with their grid block I,J,K locations. Connections must be defined in COMPDAT before they can be referenced in any other keyword.</td>
</tr>
<tr>
<td>Hysteresis option only</td>
<td>COMPIMB</td>
<td>Imbibition table numbers for well connections</td>
</tr>
<tr>
<td></td>
<td>COMPMOBI</td>
<td>Sets a constant user-defined mobility in injection well connections. This enables the injectivity to be calculated more accurately in cases where, for example, gas is injected into a large grid block initially containing oil.</td>
</tr>
<tr>
<td></td>
<td>COMPORD</td>
<td>Defines how the ordering of connections along the wellbore is determined. The default method uses the connection locations and penetration directions to trace the well track through the grid, but simpler ordering methods based on true depth or input order may be selected.</td>
</tr>
<tr>
<td>Cannot be used with the Vertical Equilibrium option.</td>
<td>COMPRP</td>
<td>Well connection saturation table end-point scaling data. This keyword may be used independently of any end-point scaling in the grid blocks, and it is typically applied to allow for the effects of partial penetration.</td>
</tr>
<tr>
<td></td>
<td>COMPLUMP</td>
<td>Lumps connections into completions for automatic workovers. The lumped connections are opened or closed together in a workover, and their combined rates are examined when determining whether a workover should be performed.</td>
</tr>
<tr>
<td></td>
<td>WDFAC</td>
<td>Sets the well D-factor (flow-dependent skin for free gas).</td>
</tr>
<tr>
<td></td>
<td>WDFACCOR</td>
<td>Calculates well connection D-factors (flow-dependent skin for free gas).</td>
</tr>
<tr>
<td></td>
<td>WELPI</td>
<td>Scales the connection transmissibility factors to give the well a specified steady-state productivity/injectivity index at the current reservoir conditions.</td>
</tr>
<tr>
<td>Restrictions</td>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>--------------</td>
<td>---------</td>
<td>-------------</td>
</tr>
<tr>
<td></td>
<td>WPIMULT</td>
<td>Multiplies the connection transmissibility factors by a specified value.</td>
</tr>
<tr>
<td></td>
<td>PICOND</td>
<td>Controls the calculation of the generalized pseudo-pressure option for well inflow in gas condensate runs.</td>
</tr>
<tr>
<td></td>
<td>WINJMULT</td>
<td>Specifies pressure-dependent injectivity multipliers. Sets a constant user-defined mobility in injection well connections. This enables the injectivity to be calculated more accurately in cases where, for example, gas is injected into a large grid block initially containing oil.</td>
</tr>
</tbody>
</table>

**ECLIPSE 100 only keywords**

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>COMPINJK</td>
<td>Sets a constant user-defined relative permeability for injection well connections. This enables the injectivity to be calculated more accurately in cases where, for example, gas is injected into a large grid block initially containing oil.</td>
</tr>
<tr>
<td>COMPVE</td>
<td>Well connection depth data. This keyword may be used in vertical equilibrium runs only, to define where a connection penetrates only part of the thickness of a grid block.</td>
</tr>
<tr>
<td>WPITAB</td>
<td>Assigns PI multiplier tables (input with keyword PIMULTAB) to wells.</td>
</tr>
<tr>
<td>PIMULTAB</td>
<td>Inputs a set of PI multiplier tables, which scale well connection factors as a function of the well’s maximum water cut.</td>
</tr>
<tr>
<td>WBOREVOL</td>
<td>Sets the wellbore volume, for modeling wellbore storage in well tests.</td>
</tr>
</tbody>
</table>

**ECLIPSE 300 only keywords**

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>COMPKRI</td>
<td>Sets constant user-defined relative permeabilities for each phase in injection well connections. This enables the injectivity to be calculated more accurately in cases where, for example, gas is injected into a large grid block initially containing oil.</td>
</tr>
<tr>
<td>COMPAGH</td>
<td>Sets a constant density term used for the calculation of the connection gravity head, making the gravity head independent of the density of the fluid mixture in the well bore.</td>
</tr>
<tr>
<td>PSEUPRES</td>
<td>Activates the Generalized Pseudo Pressure option for all wells.</td>
</tr>
</tbody>
</table>

**Define multisegment wells**

**Common keywords**

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>COMPSEGL</td>
<td>Defines the location of multisegment well completions in a local grid.</td>
</tr>
<tr>
<td>COMPSEGS</td>
<td>Defines the location of completions in a multisegment well.</td>
</tr>
<tr>
<td>WELSEGS</td>
<td>Defines the segment structure of a multisegment well.</td>
</tr>
<tr>
<td>WSEGAICD</td>
<td>Defines segments to represent an ‘autonomous’ inflow control device.</td>
</tr>
<tr>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>--------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>WSEGDFIN</td>
<td>Inputs a table of inclination scaling factors versus deviation angle for the Drift Flux slip model</td>
</tr>
<tr>
<td>WSEGDFMD</td>
<td>Defines the Drift Flux slip model</td>
</tr>
<tr>
<td>WSEGDFPA</td>
<td>Changes parameter values in Drift Flux slip model.</td>
</tr>
<tr>
<td>WSEGEXSS</td>
<td>Specifies fluid import/removal through an external source/sink.</td>
</tr>
<tr>
<td>WSEGFLIM</td>
<td>Defines segments to represent a flow limiting valve.</td>
</tr>
<tr>
<td>WSEGFMOD</td>
<td>Specifies the multi-phase flow model and related parameters.</td>
</tr>
<tr>
<td>WSEGLABY</td>
<td>Defines segments to represent a ‘labyrinth’ inflow control device.</td>
</tr>
<tr>
<td>WSEGLINK</td>
<td>Defines the chord segment links for specifying looped flow paths.</td>
</tr>
<tr>
<td>WSEGMULT</td>
<td>Imposes constant or variable multipliers for segment frictional pressure drops.</td>
</tr>
<tr>
<td>WSEGPROP</td>
<td>Modifies the properties of individual well segments.</td>
</tr>
<tr>
<td>WSEGSAICD</td>
<td>Defines segments to represent a ‘spiral’ inflow control device.</td>
</tr>
<tr>
<td>WSEGSTABL</td>
<td>Designates segments to have their pressure drops calculated from a VFP table.</td>
</tr>
<tr>
<td>WSEGVALV</td>
<td>Defines segments to represent a sub-critical valve.</td>
</tr>
</tbody>
</table>

**ECLIPSE 100 only keywords**

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>WFRICSEG</td>
<td>Converts WFRICITN keyword data for a friction well into data for a multisegment well.</td>
</tr>
<tr>
<td>WFRICSGL</td>
<td>Converts WFRICITNL keyword data for a friction well into data for a multisegment well.</td>
</tr>
<tr>
<td>WSEGINIT</td>
<td>Sets initial pressure and fluid contents of well segments.</td>
</tr>
<tr>
<td>WSEGITER</td>
<td>Activates the robust iteration scheme for solving multisegment wells.</td>
</tr>
<tr>
<td>WSEGPULL</td>
<td>Defines a pull through pump for a down-hole water separator.</td>
</tr>
<tr>
<td>WSEGSEP</td>
<td>Defines a segment to represent a down-hole separator.</td>
</tr>
</tbody>
</table>

**ECLIPSE 300 only keywords**

<table>
<thead>
<tr>
<th>Restrictions</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal only</td>
<td>WSEGHEAT</td>
<td>Specifies heat loss from a segment.</td>
</tr>
<tr>
<td></td>
<td>WSEGWELL</td>
<td>Associates a segment of a multisegment well with a secondary well name</td>
</tr>
</tbody>
</table>
Define separators (ECLIPSE 300)

- **SEPCOND** Defines the conditions of a named separator, and assigns this as the default separator for all wells subordinate to a particular group.

- **WSEPCOND** Assigns named separators to wells, in place of their group’s default separator specified in SEPCOND

The separator conditions associated with a well define the splitting of the well production in to stock tank oil and gas.

![Separator Diagram](image)

**Figure 2.1. Splitting well production into stock tank oil and gas**

Separator conditions are defined using the SEPCOND keyword. This sets the pressure and temperature for each separator stage, the method to be used to flash the stage, and the coupling to other stages. The last stage is always used to describe the stock tank. For a separator train consisting of two stages and a stock tank, the SEPCOND keyword required would be:

```
SEPCOND
SEP1 Field 1  65 815 /
SEP1 Field 2  65  65 /
SEP1 Field 3  45 14.7 /
/
```

Note that a final stock tank step is **not** supplied automatically. In this case the separator conditions are SEP1. The first stage has a temperature of 65 °F and a pressure of 815 psi, the second a temperature of 65 °F and a pressure of 65 psi, and the stock tank is at 45 °F and 14.7 psi. The coupling is the default: the liquid from stage 1 feeding stage 2, and the liquid from stage 2 feeding the stock tank. Note that all separators in a given group should have the same final stage (stock tank) conditions, in order that oil volumes are added consistently. A separator stage may be re-specified, or the whole separator condition re-specified. The separator stages must, however, be introduced in increasing order, in order to prevent missing stages when the simulation starts.

**Well controls and limits**

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>WEFAC</strong></td>
<td>Sets well efficiency factors, to take account of downtime when wells are taken down individually on a regular basis.</td>
</tr>
<tr>
<td><strong>WELOPEN</strong></td>
<td>Opens or closes wells or connections.</td>
</tr>
<tr>
<td><strong>WLIST</strong></td>
<td>Constructs well lists for use with other well keywords.</td>
</tr>
</tbody>
</table>
Production well control data

Common keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Either WCONPROD or WCONHIST should be used to specify production wells</td>
<td>WCONPROD</td>
<td>Control data for production wells.</td>
</tr>
<tr>
<td>once they have been introduced with WELSPECS and COMPDAT.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Either WCONPROD or WCONHIST should be used to specify production wells</td>
<td>WCONHIST</td>
<td>Observed production rates for history matching wells.</td>
</tr>
<tr>
<td>once they have been introduced with WELSPECS and COMPDAT.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>WHISTCTL</td>
<td></td>
<td>Sets control mode instructions for all history matching wells.</td>
</tr>
<tr>
<td>WELTARG</td>
<td></td>
<td>Resets an individual operating target or limit.</td>
</tr>
<tr>
<td>WELCNTL</td>
<td></td>
<td>Resets the control mode and operating target on wells.</td>
</tr>
<tr>
<td>WELDRAW</td>
<td></td>
<td>Sets a maximum allowable drawdown for production wells.</td>
</tr>
<tr>
<td>WTADD</td>
<td></td>
<td>Adds to a well target or limit by a given amount.</td>
</tr>
<tr>
<td>WTMULT</td>
<td></td>
<td>Multiplies a well target or limit by a given factor.</td>
</tr>
<tr>
<td>WLISTARG</td>
<td></td>
<td>Resets individual operating targets or limits for a list of wells, constructed with the WLIST keyword.</td>
</tr>
<tr>
<td>WREGROUP</td>
<td></td>
<td>Switches wells automatically between groups as certain limits are violated.</td>
</tr>
<tr>
<td>LINCOM</td>
<td></td>
<td>Specifies coefficients for a linear combination of oil, water and gas.</td>
</tr>
</tbody>
</table>

ECLIPSE 100 only keywords

- **WGORPEN** Sets data for the GOR Penalty option

Injection well control data

Common keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>WCONINJE</td>
<td></td>
<td>Control data for injection wells. Either WCONINJE or WCONINJH should be used to specify injection wells once they have been introduced with WELSPECS and COMPDAT.</td>
</tr>
<tr>
<td>WCONINJH</td>
<td></td>
<td>Observed injection rates for history matching wells. be used to specify injection wells once they have been introduced with WELSPECS and COMPDAT.</td>
</tr>
<tr>
<td>Restriction</td>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>------------------</td>
<td>-------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td></td>
<td>WCONINJP</td>
<td>Sets the rate of an injection well in terms of the voidage of the production wells that surround it.</td>
</tr>
<tr>
<td></td>
<td>WHISTCTL</td>
<td>Sets control mode instructions for all history matching wells.</td>
</tr>
<tr>
<td></td>
<td>WELTARG</td>
<td>Resets an individual operating target or limit.</td>
</tr>
<tr>
<td></td>
<td>WELCNTLT</td>
<td>Resets the control mode and operating target on wells</td>
</tr>
<tr>
<td></td>
<td>WTMULT</td>
<td>Multiplies a well target or limit by a given factor.</td>
</tr>
<tr>
<td></td>
<td>WLISTARG</td>
<td>Resets individual operating targets or limits for a list of wells, constructed with the WLIST keyword.</td>
</tr>
<tr>
<td>Tracer option only</td>
<td>WTRACER</td>
<td>Defines the concentration of tracers in the well injection streams</td>
</tr>
<tr>
<td>Brine option only</td>
<td>WSALT</td>
<td>Defines the salt concentration in water injectors</td>
</tr>
</tbody>
</table>

### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature option only</td>
<td>WAPI</td>
<td>Defines the API of injected oil in API tracking cases.</td>
</tr>
<tr>
<td></td>
<td>WTEMP</td>
<td>Defines the temperature of injected water</td>
</tr>
</tbody>
</table>

### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal</td>
<td>WINJGAS</td>
<td>Defines the composition of the fluid injected by wells.</td>
</tr>
<tr>
<td></td>
<td>WINJOIL</td>
<td>Defines the composition of the fluid injected by wells.</td>
</tr>
<tr>
<td>Water component model</td>
<td>WINJTEMP</td>
<td>Defines the temperature of the fluid injected by wells.</td>
</tr>
<tr>
<td></td>
<td>WINJW</td>
<td>Defines the composition of the water injected by wells</td>
</tr>
<tr>
<td></td>
<td>WELLSTRE</td>
<td>Sets the composition of a named wellstream for injection.</td>
</tr>
<tr>
<td>Water component model</td>
<td>WELLSTRW</td>
<td>Sets the composition of a named wellstream for water injection</td>
</tr>
<tr>
<td></td>
<td>WINJEDET</td>
<td>Describes detailed composition of a lumped fluid for an injection well.</td>
</tr>
<tr>
<td></td>
<td>WINJMIX</td>
<td>Defines an injection mixture by combining fluid from a set of sources according to specified fractions.</td>
</tr>
<tr>
<td></td>
<td>WINJORD</td>
<td>Defines an injection mixture by combining fluid from a set of sources in a specified order.</td>
</tr>
<tr>
<td></td>
<td>WELLWAG</td>
<td>Specifies WAG well injection targets.</td>
</tr>
</tbody>
</table>
Vertical flow performance keywords

Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>VFPPROD</td>
<td>Supplies a production well VFP table. VFP tables are required only if there are tubing head pressure targets or limits. The VFP table is also used to supply Tubing head temperature values (see the WHTEMP keyword). A VFP table must be entered before it is referenced in WCONPROD or WHTEMP.</td>
</tr>
<tr>
<td>VFPINJ</td>
<td>Supplies an injection well VFP table. VFP tables are required only if there are tubing head pressure targets or limits. A VFP table must be entered before it is referenced in WCONINJE.</td>
</tr>
<tr>
<td>VFPCHK</td>
<td>Turns off checks for crossing VFP curves when VFP tables are read in, or sets a threshold BHP value above which curve crossings will be ignored (for example to allow for supersonic flow conditions).</td>
</tr>
<tr>
<td>WVFPDP</td>
<td>Adds a user-defined adjustment to a well’s BHP obtained by interpolating the VFP tables (for example to help match a well’s flow rate at a given THP).</td>
</tr>
<tr>
<td>WVFPEXP</td>
<td>Instructs a well to use explicit water and gas fractions for VFP table look-up, or when using stabilized VFP tables, to be shut-in at the end of the timestep when it is operating in a part of the table that corresponds to the unstable region. Can also stop a well from hunting between THP and rate control when the rate limit forces the well to operate in the unstable part of the VFP curve.</td>
</tr>
<tr>
<td>WHTEMP</td>
<td>This keyword is used to specify the VFP table reference number for Tubing Head Temperature calculations or to set the constant Tubing Head Temperature value. This option is supported only for production wells and not for injection wells.</td>
</tr>
</tbody>
</table>

ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>VFPTABL</td>
<td>Requests cubic spline interpolation for the artificial lift quantity. This may be useful when running the Gas Lift Optimization facility, by making the gradient of the wells’ performance with reference to the lift gas injection rate vary more smoothly over the table.</td>
</tr>
<tr>
<td>WALQCALC</td>
<td>Defines the well ALQ to be set equal to the surface density of the well’s produced oil or gas.</td>
</tr>
</tbody>
</table>

ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>VFPCHECK</td>
<td>Contains data that ECLIPSE 300 uses to check for consistency between the simulator fluid model and that used in VFP i to generate the VFPPROD and/or VFPINJ keywords.</td>
</tr>
</tbody>
</table>
Well economic limits and coning control

Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>WECON</td>
<td>Economic operating limits on well rates and production ratios.</td>
</tr>
<tr>
<td>CECON</td>
<td>Economic operating limits on individual well connections.</td>
</tr>
<tr>
<td>WCUTBACK</td>
<td>Instructions to cut back well rates to limit coning.</td>
</tr>
<tr>
<td>WLIMTOL</td>
<td>Tolerance fraction for economic limits. If the limit is broken by more than the specified tolerance, the timestep will be repeated after the remedial action has been taken.</td>
</tr>
<tr>
<td>WORKTHP</td>
<td>Instructions to perform a workover if the well is unable to produce at its THP limit.</td>
</tr>
</tbody>
</table>

ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>WECONINJ</td>
<td>Sets a minimum economic rate for injection wells.</td>
</tr>
<tr>
<td>WTHPMAX</td>
<td>Sets a maximum THP design limit.</td>
</tr>
<tr>
<td>WBHGLR</td>
<td>Sets upper limits on the gas-liquid ratio at bottom hole conditions, which will trigger workovers or rate cutbacks.</td>
</tr>
<tr>
<td>CECONT</td>
<td>Economic limits on well connection tracer rates and concentrations.</td>
</tr>
<tr>
<td>WCUTBACT</td>
<td>Well rate cutbacks and reversals based on tracer rates and concentrations.</td>
</tr>
<tr>
<td>WECONT</td>
<td>Economic limits on well tracer rates and concentrations.</td>
</tr>
</tbody>
</table>

ECLIPSE 300 only keywords

• WECONCMF Economic operating limits on component mole fractions.

Group controls and limits

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NUPCOL</td>
<td>Sets the number of iterations in each timestep for which well flow targets are recalculated for group control.</td>
</tr>
<tr>
<td>GCONTROL</td>
<td>Sets tolerance fractions and iteration limits for group control targets and (in ECLIPSE 300) gas injection calculations.</td>
</tr>
<tr>
<td>WLIMTOL</td>
<td>Tolerance fraction for workover limits.</td>
</tr>
</tbody>
</table>

Group production control by guide rate

Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GCONPROD</td>
<td>Group production rate targets and limits for control by guide rate.</td>
</tr>
<tr>
<td>GRUPTARG</td>
<td>Resets an individual production rate target or limit for a group.</td>
</tr>
<tr>
<td>Keyword</td>
<td>Description</td>
</tr>
<tr>
<td>------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>GPMAINT</td>
<td>Sets group production or injection target to maintain the average pressure in a fluid-in-place region.</td>
</tr>
<tr>
<td>WGRUPCON</td>
<td>Sets or scales well guide rates for group control.</td>
</tr>
<tr>
<td>GUIDERAT</td>
<td>Coefficients for a general formula for variable well guide rates.</td>
</tr>
<tr>
<td>GTADD</td>
<td>Adds to a group rate target or limit by a given factor.</td>
</tr>
<tr>
<td>GTMULT</td>
<td>Multiplies a group rate target or limit by a given factor.</td>
</tr>
</tbody>
</table>

**ECLIPSE 100 keywords**

- **PRORDER** Defines order of applying group production rules, a sequence of actions to maintain group production at its target rate.

**ECLIPSE 300 keywords**

- **DGRDT** Limits the rate of change of guide rate calculated for wells under group control.
- **GQUALITY** Sets group gas quality target.

**Group production control by prioritization**

- **PRIORITY** Activates prioritization option and sets priority coefficients
- **GCONFRI** Group production rate limits for prioritization option.
- **GRUPTARG** Resets an individual production rate limit for a group.
- **WELPRI** Sets priority numbers for wells.

**Group economic limits and coning control**

**Common keywords**

- **GECON** Economic operating limits on group rates and production ratios.

**ECLIPSE 100 only keywords**

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GCUTBACK</td>
<td>Instructions to cut back group rates to limit water/gas breakthrough.</td>
</tr>
<tr>
<td>GCUTBACT</td>
<td>Instructions to cut back group rates based on tracer rates and concentrations.</td>
</tr>
<tr>
<td>GECONT</td>
<td>Economic limits on group tracer rates and concentrations.</td>
</tr>
</tbody>
</table>

**Group injection control**

**Common keywords**

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GCONINJE</td>
<td>Group injection rate targets and limits.</td>
</tr>
</tbody>
</table>
### Keyword | Description
--- | ---
| **GPMAINT** | Sets group production or injection target to maintain the average pressure in a fluid-in-place region. |
| **WGRUPCON** | Well guide rates for group control. |

### ECLIPSE 100 only keywords
- **GCONSUMP** Gas consumption and import rates.

### ECLIPSE 300 only keywords
| ECLIPSE 300 only keyword | Description |
--- | ---
| **GINJGAS** | Defines the composition of the fluid injected by groups. |
| **WELLESTRE** | Sets the composition of a named wellstream for injection. |
| **WINJMJX** | Defines an injection mixture by combining fluid from a set of sources according to specified fractions. |
| **WINJORD** | Defines an injection mixture by combining fluid from a set of sources in a specified order. |
| **GADVANCE** | Assigns advance import gas to a group. |
| **GRUPFUEL** | Specifies fuel gas usage rate for a group. |
| **GRUPSALE** | Specifies gas sales rate for a group. |
| **WTAKEGAS** | Specifies the order in which fuel, sales and reinjection gas are extracted from the production gas stream of a source group. |
| **WAVAILIM** | Requests that gas reinjection rates should take account of the availability of gas supply. |

### Group efficiency factor
- **GEFAC** Sets group efficiency factors, to take account of downtime when all the group’s subordinate wells have their downtimes synchronized.

### Sales gas control (ECLIPSE 100)
This facility provides a means of controlling the export of associated gas from an oil field, which may at the same time have controls on the oil production rate. The gas export rate is controlled by reinjecting the surplus gas not required for sale.
- **GCONSOLE** Sales gas production targets and limits for groups and the field.
- **GCONSUMP** Gas consumption and import rates.
- **WGRUPCON** Well guide rates for group control.
- **WGASPROD** Defines wells as special gas producers for sales gas control.
Satellite group flows

Satellite groups can be used to incorporate known production and injection flows from outside the simulation grid. They are groups that contain no wells or subordinate groups, and whose production and injection rates may be set periodically with the following keywords.

Common keywords

- **GSATPROD** Production rate data for satellite groups
- **GSATINJE** Injection rate data for satellite groups.

ECLIPSE 300 only keywords

- **GSATCOMP** Fluid composition for satellite groups.

Sustainable capacity test

- **GSSCPTST** This allows the production rate which a group of wells is capable of maintaining for a period of time to be determined.

Automatic drilling and workover controls

Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>WDRILTIM</td>
<td>Time taken to drill each well. Limits the rate at which new wells can be drilled.</td>
</tr>
<tr>
<td>QDRILL</td>
<td>Places wells in the sequential drilling queue. Wells are opened from this drilling queue in the sequence in which they were placed in the queue, whenever they are needed to maintain a group rate target under group control by guide rate (GCONPROD, GCONINJE, GCONSALE) or when needed to maintain a group’s production potential (GDRILPOT).</td>
</tr>
<tr>
<td>WDRILPRI</td>
<td>Places wells in the prioritized drilling queue. This operates in a similar way to the sequential drilling queue (keyword QDRILL) except that wells are opened in decreasing order of drilling priority. Well drilling priorities can either be set manually or calculated as a function of their potential rates according to a formula specified with the DRILPRI keyword.</td>
</tr>
<tr>
<td>DRILPRI</td>
<td>Defines the default priority formula for the prioritized drilling queue set up with keyword WDRILPRI.</td>
</tr>
<tr>
<td>GDRILPOT</td>
<td>Causes wells to be drilled from the drilling queue when a group’s production rate potential falls below a specified value.</td>
</tr>
<tr>
<td>WORKLIM</td>
<td>Time taken to perform a workover. Limits the rate of doing workovers.</td>
</tr>
<tr>
<td>GRUPRIG</td>
<td>Assigns workover rigs to groups. Together with WORKLIM, it limits the rate at which workovers can be performed within particular groups.</td>
</tr>
<tr>
<td>COMPLUMP</td>
<td>Lumps connections into completions for automatic workovers. The lumped connections will be opened or closed together in a workover, and their combined rates will be examined when determining whether a workover should be performed.</td>
</tr>
</tbody>
</table>
## ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>WDRILRES</td>
<td>Prevents two wells being drilled in the same grid block</td>
</tr>
<tr>
<td>WELSOMIN</td>
<td>Prevents new connections being opened in grid blocks having less than a specified minimum oil saturation.</td>
</tr>
<tr>
<td>PRORDER</td>
<td>Defines order of applying group production rules, including drilling and re-perforating, to maintain group production at its target rate.</td>
</tr>
<tr>
<td>WPLUG</td>
<td>Well plug-back lengths, for automatic plug-backs.</td>
</tr>
</tbody>
</table>

### Action keyword facility

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACTION</td>
<td>Initiates a set of actions (defined by SCHEDULE section keywords) to be executed when a specified field condition is satisfied.</td>
</tr>
<tr>
<td>ACTIONG</td>
<td>Defines actions to be triggered by group quantities.</td>
</tr>
<tr>
<td>ACTIONR</td>
<td>Defines actions to be triggered by region quantities.</td>
</tr>
<tr>
<td>ACTIONW</td>
<td>Defines actions to be triggered by well quantities.</td>
</tr>
<tr>
<td>ACTIONS</td>
<td>Defines actions to be triggered by well segment quantities.</td>
</tr>
<tr>
<td>ACTIONX</td>
<td>Defines actions to be triggered by a set of conditions on various quantities</td>
</tr>
<tr>
<td>DELAYACT</td>
<td>Defines actions to be triggered a given time after another action has been triggered.</td>
</tr>
<tr>
<td>ENDACTION</td>
<td>Indicates the end of the set of actions initiated with the ACTION keyword family.</td>
</tr>
</tbody>
</table>

### Periodic testing and cycling wells

#### Common keywords

- **WTEST** Instructions for periodic testing of closed wells, to see if they can operate again.

#### ECLIPSE 100 only keywords

- **WCYCLE** Instructions for automatically cycling wells on and off.

#### ECLIPSE 300 only keywords

- **WELLWAG** Specifies WAG well injection targets.

### Artificial lift operations (ECLIPSE 100)

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>WLIFF</td>
<td>Instructions for automatic retubing, lift switching and THP reduction, when a well or group production rate falls below a specified value.</td>
</tr>
</tbody>
</table>
### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GLIFTLIM</td>
<td>Limits on artificial lift capacity for groups. Prevents further automatic lift switching when group lift capacity would be exceeded.</td>
</tr>
<tr>
<td>WLIMTOL</td>
<td>Tolerance fraction for lift switching limits. If violated by more than the tolerance, the timestep will be repeated after retubing or lift switching.</td>
</tr>
<tr>
<td>WELTARG</td>
<td>Change VFP table number or artificial lift quantity, for manual retubing or lift switching.</td>
</tr>
<tr>
<td>PRORDER</td>
<td>Defines order of applying group production rules, including artificial lift operations, to maintain group production at its target rate.</td>
</tr>
<tr>
<td>VFPTABL</td>
<td>Requests cubic spline interpolation for the artificial lift quantity. This may be useful when running the Gas Lift Optimization facility, by making the gradient of the wells’ performance with reference to the lift gas injection rate vary more smoothly over the table.</td>
</tr>
</tbody>
</table>

### Scale deposition model

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SCDATAB</td>
<td>Defines scale damage tables which govern the reduction in the connection transmissibility factor caused by deposited scale.</td>
</tr>
<tr>
<td>SCDPTAB</td>
<td>Defines scale deposition tables which govern the rate of scale deposition (per unit water flow) at a well connection. This rate depends on the fraction of sea water present in the water flowing through the connection.</td>
</tr>
<tr>
<td>SCDPTRAC</td>
<td>Sets a passive water tracer to be associated with the sea water fraction.</td>
</tr>
<tr>
<td>WSCTAB</td>
<td>Specifies which scale deposition table and which scale damage table will be used to model scale effects for a well.</td>
</tr>
<tr>
<td>WSCCLEAN</td>
<td>Reduces the amount of scale deposited around well connections</td>
</tr>
<tr>
<td>WSCCLENL</td>
<td>Reduces the amount of scale deposited around local grid well connections</td>
</tr>
</tbody>
</table>

### Flash transformation and separator conditions

#### Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>COMPFLSH</td>
<td>Sets flash transformation ratios to apply to the well connection flows. They can be used to transform oil and gas production rates from differential liberation conditions to flash liberation conditions.</td>
</tr>
</tbody>
</table>

#### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>GSEPCOND</td>
<td>Assigns separators to groups, for use with SEPVALS</td>
</tr>
</tbody>
</table>
Use of SEPVALS is not allowed in runs using the COMPFLUSH keyword.

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Use of SEPVALS is not allowed in runs using the COMPFLUSH keyword.</td>
<td>SEPVALS</td>
<td>Enters separator test results. The keyword can be used firstly to define the initial separator conditions, and subsequently to change them. When the separator conditions have been changed, ECLIPSE will transform the oil and gas flow rates of the wells and groups that feed into the separator.</td>
</tr>
</tbody>
</table>

### Gas processing (ECLIPSE 300)

- **GPTABLE** Tables of gas plant recovery factors
- **GPTABLEN** Gas plant table with NGL recoveries
- **GPTABLE3** Gas plant table with oil, NGL and gas recoveries
- **RECOVERY** Recovery plant table.

### Aquifer controls

#### Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AQUFLUX</td>
<td>Modifies the water influx rate from a constant flux aquifer.</td>
</tr>
<tr>
<td>AQUCHGAS</td>
<td>Modifies the gas head for a constant head gas aquifer.</td>
</tr>
<tr>
<td>AQUCHWAT</td>
<td>Modifies the water head for a constant head water aquifer.</td>
</tr>
<tr>
<td>AQUIT</td>
<td>Modifies the properties of a Carter-Tracy aquifer. AQUFETP</td>
</tr>
</tbody>
</table>

#### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RAINFALL</td>
<td>Specifies a monthly flux rate for constant flux aquifers.</td>
</tr>
<tr>
<td>AQUCWFAILC</td>
<td>Modifies the properties of all AQUCHWAT type aquifers.</td>
</tr>
</tbody>
</table>

#### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AQSTREAM</td>
<td>Sets composition of gas stream for AQUCHGAS aquifer. For the GASWAT and CO2STORE options, sets composition of water AQUCHWAT, AQUCT, AQUFETP and AQUFLUX aquifers.</td>
</tr>
<tr>
<td>AQSTREAM</td>
<td>For the multi-component water option (COMPW), sets composition of water AQUCHWAT, AQUCT, AQUFETP and AQUFLUX aquifers.</td>
</tr>
</tbody>
</table>
Dissolved gas and vaporized oil constraints (ECLIPSE 100)

<table>
<thead>
<tr>
<th>Restriction</th>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>If not specified, gas redissolves instantly in undersaturated oil</td>
<td>DRSDT</td>
<td>Gas re-solution rate.</td>
</tr>
<tr>
<td>If not specified, oil vaporizes instantly in undersaturated wet gas</td>
<td>DRVDT</td>
<td>Oil vaporization rate</td>
</tr>
<tr>
<td>Only for systems with both wet gas and live oil</td>
<td>VAPPARS</td>
<td>Parameters controlling vaporization of oil in under-saturated gas.</td>
</tr>
</tbody>
</table>

Geomechanics option (ECLIPSE 300)

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DISPBC</td>
<td>Permits the application of displacement (Dirichlet) or velocity boundary conditions.</td>
</tr>
<tr>
<td>GMDISBC</td>
<td>Permits the application of displacement (Dirichlet) boundary conditions.</td>
</tr>
<tr>
<td>GMPSTBC</td>
<td>Principal stresses can be applied as boundary conditions.</td>
</tr>
<tr>
<td>GMTRABC</td>
<td>Permits application of traction (Neumann) boundary conditions.</td>
</tr>
<tr>
<td>PSTRBC</td>
<td>Principal stresses can be applied as boundary conditions.</td>
</tr>
<tr>
<td>STRESBC</td>
<td>Permits the application of a load or a velocity (rigid) boundary condition on all external and internal surfaces of the simulation grid.</td>
</tr>
<tr>
<td>TRACTBC</td>
<td>Permits application of traction (Neumann) boundary conditions.</td>
</tr>
<tr>
<td>WSCTAB</td>
<td>Links a well to a scale damage table.</td>
</tr>
</tbody>
</table>

Grid block property modifications

Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BOX</td>
<td>Defines the current input box, for use with MULTX, MULTX-, MULTY, MULTY-, MULTZ, MULTZ- and MULTPV.</td>
</tr>
<tr>
<td>MULTFLT</td>
<td>Modifies the transmissibility across a named fault.</td>
</tr>
<tr>
<td>MULTPV</td>
<td>Multiplies the pore volumes of cells within the current input box.</td>
</tr>
<tr>
<td>MULTX</td>
<td>Multiplies X-direction transmissibilities of cells within the current input box</td>
</tr>
<tr>
<td>MULTX-</td>
<td>Multiplies negative X-direction transmissibilities of cells within the current input box</td>
</tr>
<tr>
<td>MULTY</td>
<td>Multiplies Y-direction transmissibilities of cells within the current input box</td>
</tr>
<tr>
<td>MULTY-</td>
<td>Multiplies negative Y-direction transmissibilities of cells within the current input box</td>
</tr>
<tr>
<td>MULTZ</td>
<td>Multiplies Z-direction transmissibilities of cells within the current input box</td>
</tr>
</tbody>
</table>
### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>EXCAVATE</td>
<td>Removes cells during a run.</td>
</tr>
</tbody>
</table>

### ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>OVERTR</td>
<td>Overwrites R direction transmissibilities.</td>
</tr>
<tr>
<td>OVERTTHT</td>
<td>Overwrites theta direction transmissibilities.</td>
</tr>
<tr>
<td>OVERTX</td>
<td>Overwrites X direction transmissibilities.</td>
</tr>
<tr>
<td>OVERTY</td>
<td>Overwrites Y direction transmissibilities.</td>
</tr>
<tr>
<td>OVERTZ</td>
<td>Overwrites Z direction transmissibilities.</td>
</tr>
</tbody>
</table>

### Dynamic Regions Facility

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DYNAMICR</td>
<td>Defines regions based on when a set of conditions are satisfied. These regions may be static or allowed to change dynamically.</td>
</tr>
<tr>
<td>ENDDYN</td>
<td>Indicates the end of a DYNAMICR keyword sequence</td>
</tr>
</tbody>
</table>

### Dual porosity

- **DIFFMMF** Diffusivity multipliers for matrix-fracture diffusive coupling.
- **MULSGGD** Specify dual porosity alternative oil-gas matrix-fracture transmissibility multiplier
- **MULSGGDV** Specify dual porosity alternative oil-gas matrix-fracture transmissibility multipliers
- **MULTSIG** Specify dual porosity matrix-fracture transmissibility multiplier
- **MULTSIGV** Specify dual porosity matrix-fracture transmissibility multipliers
Lagrangian Tracer Solver (ECLIPSE 300)

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LAGTRUPD</td>
<td>Defines the streamline update conditions and solver method for the Lagrangian tracer solver.</td>
</tr>
<tr>
<td>LTRACE</td>
<td>Forces a streamline trace at the end of the next timestep.</td>
</tr>
<tr>
<td>NOLTRACE</td>
<td>Prevents streamline tracing until the next LTRACE or LAGTRUPD.</td>
</tr>
<tr>
<td>RPTSLN</td>
<td>Controls the output of streamline data as SLN files.</td>
</tr>
</tbody>
</table>

Simulator controls

The keywords are:

Common keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NEXTSTEP (or NEXT)</td>
<td>Restricts the next timestep to a specified maximum size.</td>
</tr>
<tr>
<td>TUNING</td>
<td>Timestep and convergence controls.</td>
</tr>
<tr>
<td>SKIPREST</td>
<td>Makes ECLIPSE skip the subsequent keywords in a restart run until the restart time is reached.</td>
</tr>
<tr>
<td>NSTACK</td>
<td>Sets the linear solver stack size.</td>
</tr>
<tr>
<td>ZIPPY2</td>
<td>Experimental feature to automatically select timesteps in poorly converging runs.</td>
</tr>
<tr>
<td>ZIPP2OFF</td>
<td>Turns off ZIPPY2 feature.</td>
</tr>
</tbody>
</table>

ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TUNINGDP</td>
<td>Supplies a modified set of convergence parameters for some high throughput cases.</td>
</tr>
<tr>
<td>TIGHTEN</td>
<td>Enables convergence parameters to be tightened</td>
</tr>
<tr>
<td>TIGHTENP</td>
<td>Enables convergence parameters to be tightened.</td>
</tr>
<tr>
<td>DIMPES</td>
<td>Sets the IMPES solution procedure.</td>
</tr>
<tr>
<td>DIMPLECT</td>
<td>Restores the fully implicit solution procedure.</td>
</tr>
<tr>
<td>MATCORR</td>
<td>Activates the material balance correction option, to reduce the accumulated material balance error.</td>
</tr>
<tr>
<td>READDATA</td>
<td>Reads files of SCHEDULE data that may be constructed externally during the course of the run.</td>
</tr>
<tr>
<td>OPTIONS</td>
<td>Activate special options within particular facilities in ECLIPSE 100.</td>
</tr>
</tbody>
</table>
## ECLIPSE 300 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TOPT</td>
<td>Preset tuning options.</td>
</tr>
<tr>
<td>TSCRIT</td>
<td>Timestep controls.</td>
</tr>
<tr>
<td>CVCRIT</td>
<td>Convergence criteria.</td>
</tr>
<tr>
<td>FCCRIT</td>
<td>Flash convergence criteria.</td>
</tr>
<tr>
<td>LSCRIT</td>
<td>Linear solution convergence criteria.</td>
</tr>
<tr>
<td>LSCRITS</td>
<td>Linear solution convergence criteria for each individual sub-grids.</td>
</tr>
<tr>
<td>AIMFRAC</td>
<td>Sets the target implicit fraction in AIM mode.</td>
</tr>
<tr>
<td>AIMPVI</td>
<td>Specifies the minimum pore volume for an explicit cell in AIM mode.</td>
</tr>
<tr>
<td>AIMCON</td>
<td>Controls implicit cells within an AIM run.</td>
</tr>
<tr>
<td>CHANDIMS</td>
<td>Increase well array dimensions.</td>
</tr>
<tr>
<td>OPTIONS3</td>
<td>Activate special options within particular facilities in ECLIPSE 300.</td>
</tr>
<tr>
<td>WCOMPRES</td>
<td>Reduce the memory requirements of a simulation that contains a large number of wells, some of which have many connections, but most have relatively few connections</td>
</tr>
</tbody>
</table>

### User defined values

- **UDQ** Specify user defined quantities.

### Output

#### Common keywords

<table>
<thead>
<tr>
<th>Keywords</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RPTSCHED</td>
<td>Report switches to select which simulation results are to be printed at report times.</td>
</tr>
<tr>
<td>RPTST</td>
<td>Controls data written to the Restart file.</td>
</tr>
<tr>
<td>SAVE</td>
<td>Controls solution output for plotting and flexible restarts.</td>
</tr>
<tr>
<td>OUTSOL</td>
<td>Writes the current solution to a Save file for fast restarts.</td>
</tr>
<tr>
<td>BOUNDARY</td>
<td>Defines section of grid for which output maps are required.</td>
</tr>
<tr>
<td>WPAVE</td>
<td>Controls the calculation of well block average pressures that may be output to the Summary file.</td>
</tr>
<tr>
<td>WWPAVE</td>
<td>Controls the calculation of well block average pressures for an individual well. Average pressures may be output to the Summary file.</td>
</tr>
<tr>
<td>WPAVEDEP</td>
<td>Specifies a reference depth for the calculation of well block average pressures.</td>
</tr>
<tr>
<td>WRFTPLT</td>
<td>Requests output of RFT, PLT and segment data to the RFT file.</td>
</tr>
<tr>
<td>WELDEBUG</td>
<td>Sets debug output switches for individual wells. (This is primarily intended as an aid to program development and debugging, and is not recommended for general use.)</td>
</tr>
<tr>
<td>Keywords</td>
<td>Description</td>
</tr>
<tr>
<td>------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>WELEVNT</td>
<td>Sets SUMMARY keyword WPWEM output for individual wells.</td>
</tr>
<tr>
<td>WTEMPQ</td>
<td>Queries the well name and well list template matching algorithm</td>
</tr>
<tr>
<td>RPTONLY</td>
<td>Requests that the summary data is only produced at report times. The default, if RPTONLY is not present, is to write the summary information at every time.</td>
</tr>
<tr>
<td>RPTONLYO</td>
<td>Requests summary data is written every timestep. This is the default situation and the keyword need only be used to turn off a previously entered RPTONLY instruction.</td>
</tr>
<tr>
<td>RPRKPAME</td>
<td>Outputs the rock compaction</td>
</tr>
<tr>
<td>SUMTHIN</td>
<td>Reduces the number of data written to summary output files.</td>
</tr>
</tbody>
</table>

**ECLIPSE 100 only keywords**

- **WRFT** Requests output of well block data to the RFT file.

**ECLIPSE 300 only keywords**

<table>
<thead>
<tr>
<th>ECLIPSE 300 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DBGODD3P</td>
<td>Debug output for ODD3P option.</td>
</tr>
<tr>
<td>RPTPRINT</td>
<td>Controls the output of printed simulation results.</td>
</tr>
<tr>
<td>AUTOSAVE</td>
<td>Writes the solution to a Save file every ‘n’ steps.</td>
</tr>
<tr>
<td>SAVEEND</td>
<td>Requests that the solution is written to a Save file whenever the run ends.</td>
</tr>
<tr>
<td>GSEPREEPT</td>
<td>Specifies the separator chain used for sum-then-separate summary section keywords such as FGPRG</td>
</tr>
<tr>
<td>ROCKPAME</td>
<td>pore volume and the permeability multipliers versus pressure for a given cell.</td>
</tr>
</tbody>
</table>

**Conductive Fractures (ECLIPSE 300)**

- **WELLCF** Defines well behavior with conductive fractures.

**Adjoint gradient screening option (ECLIPSE 300)**

- **AJGPARAM** Specify static parameters for the adjoint gradient screening option.
- **AJGWELLS** Specify objective functions for the adjoint gradient screening option.

**Advance simulation to new time/date**

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>TSTEP</td>
<td>Advances the simulation over specified intervals between reporting times.</td>
</tr>
<tr>
<td>DATES</td>
<td>Advances the simulation to specified dates at which reports are required.</td>
</tr>
<tr>
<td>SIMULATE</td>
<td>Counteracts a NOSIM keyword and turns on the timestep simulation.</td>
</tr>
</tbody>
</table>
Keyword | Description
---|---
**TIME** | Advances the simulation to specified times at which reports are required, with respect to the **START** date in **RUNSPEC**.

**NOSIM** | Turns off simulation, but continues reading, processing and checking the subsequent keyword data.

**SIMULATE** | Counteracts a **NOSIM** keyword and turns on the timestep simulation.

**END** | Terminates the simulation.

### Instantaneous gradient option
- **GWRTWCV** Specify gradient parameters in terms of well collections.
- **GUPFREQ** Specify update frequency of the instantaneous gradient option.

### Polymer flood model

#### ECLIPSE 100 only keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100 only keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PLYADS</td>
<td>Polymer adsorption functions (tabulated).</td>
</tr>
<tr>
<td>PLYDHFLF</td>
<td>Polymer thermal degradation half life.</td>
</tr>
<tr>
<td>PLYMAX</td>
<td>Polymer/salt concentrations for mixing calculations.</td>
</tr>
<tr>
<td>PLYROCKM</td>
<td>Polymer-rock properties.</td>
</tr>
<tr>
<td>PLYSHEAR</td>
<td>Polymer shear thinning/thickening data.</td>
</tr>
<tr>
<td>PLYSHLOG</td>
<td>Activates a logarithm-based polymer shear thinning/thickening model that may be used as an alternative to <strong>PLYSHEAR</strong>. By default, this model is based on water velocity; if used in conjunction with keyword <strong>SHRATE</strong> then a switch to shear rate form is applied.</td>
</tr>
<tr>
<td>SHRATE</td>
<td>Activates the shear rate form of the alternative polymer shear thinning/thickening logarithmic model.</td>
</tr>
</tbody>
</table>

#### ECLIPSE 300 only keywords
- **PLYVISC** Polymer solution viscosity function.
- **PLYVISCS** Polymer/salt solution viscosity function.

### OPTIMIZE section

**Note:** The **OPTIMIZE** section is for use with ECLIPSE 300 only.

The **OPTIMIZE** section header must **not** be specified in an included file.
The **OPTIMIZE** section is used to define the main components of an optimization loop:

- The objective function
- The parameters to vary to maximize the objective function
- Constraints to observe during the optimization
- Optimization tuning parameters, and reporting options.

The use of the above specifications is described in "Optimization" in the *ECLIPSE Technical Description*.

**OPTIMIZE section keywords**

A list of all keywords that may occur in the **OPTIMIZE** section is given below, together with a brief description of their function. A more detailed description may be found by referring to the keyword description.

Keywords must start in column 1. All characters up to column 8 are significant.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>OPTCONS</td>
<td>Defines constraints for the reservoir optimization problem.</td>
</tr>
<tr>
<td>OPTDIMS</td>
<td>Specifies dimensions for the reservoir optimization problem.</td>
</tr>
<tr>
<td>OPTFUNC</td>
<td>Specifies the objective function for reservoir optimization.</td>
</tr>
<tr>
<td>OPTIMIZE</td>
<td>Defines the start of the optimization section.</td>
</tr>
<tr>
<td>OPTLOAD</td>
<td>Hot-starts a reservoir optimization simulation.</td>
</tr>
<tr>
<td>OPTOPTS</td>
<td>Specifies options that control the reservoir optimization.</td>
</tr>
<tr>
<td>OPTPARS</td>
<td>Defines parameters for the reservoir optimization problem</td>
</tr>
<tr>
<td>OPTTUNE</td>
<td>Defines tuning parameters for reservoir optimization.</td>
</tr>
<tr>
<td>RPTOPT</td>
<td>Reservoir optimization report options.</td>
</tr>
</tbody>
</table>

*Table 2.80: OPTIMIZE section keywords*
This section contains an alphabetic listing of keywords starting with the letter A.
ACF

Acentric factor

In a run with $N_c$ components, using an equation of state, this keyword defines the acentric factor for each component. The keyword should be followed by $N_c$ values. The use of the acentric factor in the equation of state is discussed in "Equations of State" of the ECLIPSE Technical Description.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of State Regions" of the ECLIPSE Technical Description.

DEFAULT: There are no component-wise default values. The users are recommended not to default these to the internal default value.

Examples

Example 1

ACF 0.01 0.0 0.02 0.001 /

Example 2

With three reservoir EoS regions and nine components:

ACF
0.225 0.04 0.013 0.09 0.152 0.21 0.31 0.53 0.91 /
0.225 0.04 0.013 0.09 0.152 0.23 0.30 0.54 0.89 /
0.225 0.04 0.013 0.09 0.152 0.24 0.32 0.55 0.84 /

Keywords

250
ACFDET

Acentric factor for detailed components

In a run with \( N_d \) detailed components, using an equation of state, this keyword defines the acentric factor for each detailed component. Lumped simulations are enabled with the LUMPDIMS keyword in the RUNSPEC section; further details can be found in "Delumping and lumping" in the ECLIPSE Technical Description. The keyword should be followed by \( N_d \) values. The use of the acentric factor in the equation of state is discussed in "Equations of State" in the ECLIPSE Technical Description.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of State Regions" in the ECLIPSE Technical Description.

Examples

Example 1

```
ACFDET
  0.01 0.0 0.02 0.0 0.015 0.03/
```

Example 2

With three reservoir EoS regions and nine detailed components:

```
ACFDET
  0.225 0.04 0.013 0.09 0.152 0.21 0.31 0.53 0.91 /
  0.225 0.04 0.013 0.09 0.152 0.23 0.30 0.54 0.89 /
  0.225 0.04 0.013 0.09 0.152 0.24 0.32 0.55 0.84 /
```
ACFS

Acentric factors for production system

In a run with \( N_c \) components, using an equation of state, this keyword defines the acentric factor for each component, for use in the production system. If not entered, the values will default to those specified with the ACF keyword.

The keyword should be followed by \( N_c \) values. The use of the acentric factor in the equation of state is discussed in "Equations of State" in the ECLIPSE Technical Description.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of State Regions" in the ECLIPSE Technical Description.

Example

```
ACFS
0.040 0.225 0.008 0.1255 0.2325 0.421 0.7174 0.9849 1.2737 1.6704 /
```
ACTCO2S  Activity coefficient model for carbon dioxide (CO2)

This keyword is used to specify the activity coefficient model for CO2 which is used by the CO2STORE option (salting out effect).

This keyword has one item that specifies the activity coefficient model to be used.

1. An integer selecting the model:
   0  No correction of the CO2 activity due to salts.
   1  This is the Rumpf et al. (1994) activity coefficient model with the modifications as detailed by Spycher and Pruess (2005) [Ref. 102]. It is applicable up to 160°C.
   2  This is the activity coefficient model of Duan and Sun, as modified by Spycher and Pruess (2009) [Ref. 103]. It is applicable up to 250°C.

DEFAULT: 1

Example

Selecting activity coefficient model 2 for CO2:

```
ACTCO2S
  2 /
```
Dimensions for the ACTION facility

The data consists of some or all of the following items, specifying dimensions associated with the ACTION keyword family. The data must be terminated by a slash (/).

1. The maximum number of ACTION type keywords that may be defined.
   
   DEFAULT: 2

2. The maximum number of lines of SCHEDULE data (MNLPAC), including the ENDACTIO keyword, that can be defined in any one ACTION. (This number includes comment lines and blank lines, but excludes the lines defining the triggering condition for the action.)
   
   DEFAULT: 50

3. The maximum number of characters per line of ACTION data (NCPLIN). Note that ECLIPSE rounds this number up to a multiple of 8, with a limit of 128 characters per line.
   
   DEFAULT: 80

4. The maximum number of conditions in an ACTIONX keyword.
   
   DEFAULT: 3

ECLIPSE 100 only

See keywords ACTION, ACTIONG, ACTIONR, ACTIONW, ACTIONS, ACTIONX and ENDACTIO.

Example

```
ACTDIMS
3  20  80 /
```
ACTION

Initiates a set of keywords to be processed when a specified field condition is satisfied

The ACTION keyword is one of a group of keywords that trigger actions in the simulation when certain conditions are met. While the ACTION keyword will work as documented in simulation datasets, it is recommended that you use the ACTIONX keyword for new or updated datasets as it offers the greatest functionality and flexibility.

The ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS) keyword marks the start of a set of SCHEDULE section keywords that are to be stored for later processing when a specified condition is satisfied. The ACTION keyword specifies a field condition for triggering an action. The ACTIONG, ACTIONR, ACTIONW and ACTIONS keywords offer greater flexibility, and specify actions triggered by group, region, well and segment conditions respectively. The set of keywords may be indented with one or more blank spaces for clarity and must be terminated with the ENDACTIO keyword. The keywords between the ACTION and ENDACTIO keywords are processed when the condition defined in the ACTION keyword is satisfied.

In ECLIPSE 100, almost any SCHEDULE section keyword may be placed between ACTION and ENDACTIO, with the exception of timestepping keywords (for example TIME, TSTEP, DATES and GASYEAR), the global keywords and the INCLUDE keyword. The RPTRST keyword can be used but any changes to the frequency of output will not the implemented until the report step following the satisfaction of the triggering condition.

In ECLIPSE 300, any keyword specifying data for wells, groups or separators may be placed between ACTION and ENDACTIO. The RPTRST keyword can also be used but any changes to the frequency of output will not the implemented until the report step following the satisfaction of the triggering condition.

The amount of storage space reserved for these keywords is controlled by the RUNSPEC section keyword ACTDIMS. The number of lines between an ACTION keyword and its corresponding ENDACTIO keyword (including the ENDACTIO keyword itself) is limited to MNLPAC in ACTDIMS item 2. The length of each line is limited to NCPLIN in ACTDIMS item 3.

Each action initiated with the ACTION keyword is executed only once, at the end of the timestep during which its triggering condition is first satisfied. However, any number of action keyword sets may be declared, each with its own triggering condition; the maximum number being limited to MAXACT in ACTDIMS item 1. Each action keyword set must be bracketed by a pair of ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS) and ENDACTIO keywords, and is distinguished by its action name defined in item 1 of the keyword. If an ACTION family keyword is re-entered with the same action name as a previously declared action, the action keyword set is overwritten by the new data.

The ACTION keyword contains the following items of data, terminated with a slash (/):

1. The action name.
   A name of up to 8 characters, to distinguish this action from any others.

2. The quantity to which the triggering condition applies.
   This must be one of:

   FOPR       Field oil production rate
   FWPR       Field water production rate
3. The operator for the triggering condition.
   This must be one of:
   ‘>’ Greater than
   ‘<’ Less than

4. The value of the triggering condition.
End the data record with a slash (/).

**CAUTION:** If you wish to define an action to be triggered when the field cannot meet its target flow rate, set the trigger value in item 4 to be slightly less than the target rate to allow for the fact that group/field control targets may not be met exactly (see keyword NUPCOL).

**Note:** Full checking of the keywords entered with an ACTION keyword cannot be performed until the ACTION is actually implemented, since it is possible to use wild cards in many of the keywords. However, before a simulation run is attempted the data can be run in NOSIM mode. This will then do what checks it can on the keywords within any ACTIONS that have been defined. All the most common types of ‘grammatical’ and typographical errors are found, but errors resulting from wild cards feeding inappropriate well/group names to keywords may not be found.

**CAUTION:** It is possible to nest actions by including the definition of the inner action within the keyword set of the ‘outer’ action (that is before its ENDACTIO keyword). This offers you great flexibility, but care should be exercised and the consequences of the nested actions should be thought through. For example: suppose that in the ACTDIMS keyword we have set MAXACT=1 and have then defined an action ACT1. Suppose also that as part of this action we have defined another action, ACT2. ECLIPSE only issues an error when ACT1 is actually implemented which may, of course, be well into the run. Alternatively we may call the inner action by the same name as the outer action, ACT1. This is perfectly allowable, but it must be remembered that each time an action of a given name is defined it overwrites the properties that the action may previously have had. Suppose that we have an outer action that can be repeated a number of times. Within this action is an inner one that can also be repeated and that also increments its triggering condition each time. No matter how many times the inner action has been called when the outer action is invoked again the inner action is redefined to its original specification.

### Example

This shows two action keyword sets. The first action (named ACT1) will open well PROD5 when the field gas oil ratio is greater than 3.0 and change the well oil production rate limits to 2000. The second action (named ACT2) will end the run when the average pressure in the field falls below 3000 psia.

```plaintext
ACTION
ACT1 FGOR > 3.0 /
WELOPEN
PRODS OPEN /
/
WELTARG
"P*" ORAT 2000 /
/
ENDACTIO

ACTION
ACT2 FPR < 3000.0 /
END
ENDACTIO
```
**ACTIONG**

**Initiates a set of keywords to be processed when a group satisfies a specified condition**

The ACTIONG keyword is one of a group of keywords that trigger actions in the simulation when certain conditions are met. While the ACTIONG keyword will work as documented in simulation datasets, it is recommended that you use the **ACTIONN** keyword for new or updated datasets as it offers the greatest functionality and flexibility.

The ACTIONG keyword marks the start of a set of SCHEDULE section keywords that are to be stored for later processing when a nominated group satisfies a specified condition. The set of keywords may be indented with one or more blank spaces for clarity and must be terminated with the **ENDACTIO** keyword. The keywords between the ACTIONG and ENDACTIO keywords are processed at the end of the timestep when the condition defined in the ACTIONG keyword is satisfied by a group named in item 2. You can choose whether the action is to be performed only once, or repeatedly each timestep while the triggering condition is satisfied.

If a group keyword (for example **GRUPTARG**) is placed between ACTIONG and ENDACTIO, instead of entering the name of a specific group in the keyword you may enter a question mark. ECLIPSE then applies the keyword data to whichever group (or groups) triggered the action. For example, an action can be placed on a set of groups (`G*` in item 2 of ACTIONG) to change a group’s rate limit when its water cut exceeds a certain value. The keyword GRUPTARG would be entered between ACTIONG and ENDACTIO, specifying the new rate limit and applying it to a group named `?` in item 1.

**Note:** Some keywords cannot have the group name entered as a question mark to be inherited from an ACTIONG keyword. These exceptions are those keywords where name roots are not allowed for the group name. These keywords, however, may still be used within an action if the group name is explicitly defined.

---

**ECLIPSE 100**

Practically any SCHEDULE section keyword may be placed between ACTIONG and ENDACTIO, with the exception of timestepping keywords (for example **TIME**, **TSTEP**, **DATES** and **GASYEAR**), the global keywords and the **INCLUDE** keyword. The **RPTRST** keyword can be used but any changes to the frequency of output will not the implemented until the report step following the satisfaction of the triggering condition.

**ECLIPSE 300**

Any keyword specifying data for wells, groups or separators may be placed between ACTION and ENDACTIO. The **RPTRST** keyword can also be used but any changes to the frequency of output will not be implemented until the report step following the satisfaction of the triggering condition.

The amount of storage space reserved for these keywords is controlled by the RUNSPEC section keyword **ACTDIMS**. The number of lines between an ACTIONG keyword and its corresponding ENDACTIO keyword (including the ENDACTIO keyword itself) is limited to MNLPAC in ACTDIMS item 2. The length of each line is limited to NCPLIN in ACTDIMS item 3.

Any number of action keyword sets may be declared, each with its own triggering condition; the maximum number being limited to MAXACT in ACTDIMS item 1. Each action keyword set must be bracketed by a pair of ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS) and ENDACTIO keywords, and is distinguished by its action name defined in item 1. This name must be unique across all the action keyword family. If an ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS) keyword is reentered with the same action name as a previously declared action, the previously declared action is deleted and replaced with the new action.

The ACTIONG keyword contains the following items of data, terminated with a slash (/):

---

### Keywords

258
1. The action name.
   A name of up to 8 characters, to distinguish this action from any others.
   If an action with this name has already been declared in any of the action keyword family ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS), it is overwritten with the action data specified below.

2. Group name (or name root) of the group(s) that triggers this action,
   or FIELD (for an action triggered by a field quantity).
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to make this action’s triggering condition apply to more than one group. The action will then be performed if one or more groups having this name root satisfy the triggering condition. Note that the action is performed at most once per timestep, even if more than one group satisfies the triggering condition at the end of the timestep.

3. The quantity to which the triggering condition applies.
   This must be one of:

   GOPR     Group oil production rate
   GPOT     Group oil production cumulative total
   GOIR     Group oil injection rate
   GOIT     Group oil injection cumulative total
   GWPR     Group water production rate
   GWPT     Group water production cumulative total
   GWIR     Group water injection rate
   GWIT     Group water injection cumulative total
   GGPR     Group gas production rate
   GGPT     Group gas production cumulative total
   GGIR     Group gas injection rate
   GGIT     Group gas injection cumulative total
   GVPR     Group voidage production rate
   GVPT     Group voidage production cumulative total
   GVIR     Group voidage injection rate
   GVIT     Group voidage injection cumulative total
   GLPR     Group liquid production rate
   GLPT     Group liquid production cumulative total
4. The operator for the triggering condition.
   This must be one of:

   ‘>’ Greater than

   ‘<’ Less than

5. The value of the triggering condition.
   The units depend upon the quantity selected in item 3.

6. Number of times this action can be triggered.
   The action is performed once at the end of each timestep while any group named in item 2 satisfies the triggering condition, until it has been performed the specified number of times. A number of 10,000 or greater is interpreted as ‘infinity’.

   DEFAULT: 1
7. Increment to the triggering condition.

After each time the action is performed, the specified increment is added to the triggering condition value (initialized in item 5). The increment may be negative, if desired.

DEFAULT: 0.0

End the data record with a slash (/).

**CAUTION:** If you wish to define an action to be triggered when a group cannot meet its target flow rate, set the trigger value in item 5 to be slightly less than the target rate to allow for the fact that group control targets may not be met exactly (see keyword NUPCOL).

**CAUTION:** It is possible to nest actions by including the definition of the ‘inner’ action within the keyword set of the ‘outer’ action (that is before its ENDACTIO keyword). This offers great flexibility, but care should be exercised and the consequences of the nested actions should be thought through. For example: suppose that in the ACTDIMS keyword we have set MAXACT=1 and have then defined an action ACT1. Suppose also that as part of this action we have defined another action, ACT2. ECLIPSE will only issue an error when ACT1 is actually implemented, which may, of course, be well into the run. Alternatively we may call the inner action by the same name as the outer action, ACT1. This is perfectly allowable, but it must be remembered that each time an action of a given name is defined it overwrites the properties that the action may previously have had. Suppose that we have an outer action that can be repeated a number of times. Within this action is an inner one that can also be repeated and that also increments its triggering condition each time. No matter how many times the inner action has been called when the outer action is invoked again the inner action is redefined to its original specification.

**Note:** Full checking of the keywords entered with an action keyword cannot be performed until the action is actually implemented, since it is possible to use wild cards in many of the keywords. However, before a simulation run is attempted the data can be run in NOSIM mode. This then does whatever checks it can on the keywords within any action keyword blocks that have been defined. All the most common types of ‘grammatical’ and typographical errors will be found, but errors resulting from wild cards feeding inappropriate well/group names to keywords are not found.

### Example

This shows three action keyword sets. The first action (named ACT1) applies to all groups whose names begin with G, and reduces a group’s oil rate target by 20% whenever its gas oil ratio exceeds the value of 3.0. The second action (named ACT2) opens well PROD5 when the oil production rate of group GR3 falls below 5,000. The third action (named ACT3) increases the field’s water rate limit by 50% each time the cumulative oil production increases by $1.0E6$, for a total of 3 times.

```
ACTIONG
ACT1 'G*' GGOR > 3.0 10000 /
GTMULT
'? ' ORAT 0.8 /
/
ENDACTIO
ACTIONG
ACT2 GR3 GOP' < 5000.0 /
WELOPEN
PROD5 OPEN /
/
ENDACTIO
ACTIONG
ACT3 FIELD GOPT > 1.0E6 3 1.0E6 /
```
GTMULT
FIELD WRAT 1.5 /
/
ENDACTIO
**ACTIONR**

**Initiates a set of keywords to be processed when a region satisfies a specified condition**

The ACTIONR keyword is one of a group of keywords that trigger actions in the simulation when certain conditions are met. While the ACTIONR keyword will work as documented in simulation datasets, it is recommended that you use the ACTIONX keyword for new or updated datasets as it offers the greatest functionality and flexibility.

The ACTIONR keyword marks the start of a set of SCHEDULE section keywords that are to be stored for later processing when a nominated fluid-in-place region satisfies a specified condition. The set of keywords may be indented with one or more blank spaces for clarity and must be terminated with the ENDACTIO keyword. The keywords between the ACTIONR and ENDACTIO keywords are processed at the end of the timestep when the condition defined in the ACTIONR keyword is satisfied by a region identified in items 2 and 3. You can choose whether the action is to be performed only once, or repeatedly each timestep while the triggering condition is satisfied.

In ECLIPSE 100, practically any SCHEDULE section keyword may be placed between ACTIONR and ENDACTIO, with the exception of timestepping keywords (for example TIME, TSTEP, DATES and GASYEAR), the global keywords and the INCLUDE keyword. The RPTRST keyword can be used but any changes to the frequency of output will not be implemented until the report step following the satisfaction of the triggering condition.

In ECLIPSE 300 any keyword specifying data for wells, groups or separators may be placed between ACTIONR and ENDACTIO. The RPTRST keyword can also be used but any changes to the frequency of output will not be implemented until the report step following the satisfaction of the triggering condition.

The amount of storage space reserved for these keywords is controlled by the RUNSPEC section keyword ACTDIMS. The number of lines between an ACTIONR keyword and its corresponding ENDACTIO keyword (including the ENDACTIO keyword itself) is limited to MNLPAC in ACTDIMS item 2. The length of each line is limited to NCPLIN in ACTDIMS item 3.

Any number of action keyword sets may be declared, each with its own triggering condition; the maximum number being limited to MAXACT in ACTDIMS item 1. Each action keyword set must be bracketed by a pair of ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS) and ENDACTIO keywords, and is distinguished by its action name defined in item 1. This name must be unique across all the ACTION keyword family. If an ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS) keyword is reentered with the same action name as a previously declared action, the previously declared action is deleted and replaced with the new action.

The ACTIONR keyword contains the following items of data, terminated with a slash (/)

1. The action name.
   
   A name of up to 8 characters, to distinguish this action from any others.
   
   If an action with this name has already been declared in any of the action keyword family ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS), it is overwritten with the action data specified below.

2. Fluid-in-place region number, whose conditions trigger this action.
   
   This should be an integer between 0 and NTFIP (see keyword REGDIMS in the RUNSPEC section). Region 0 refers to the whole field.
   
   Fluid-in-place regions are defined with the keyword FIPNUM in the REGIONS section (or, for ECLIPSE 100, the keyword FIP if a different family of regions is requested in item 3).
3. The fluid-in-place region family, to which the specified region belongs.
   The family name should be up to 5 characters long.
   A default (1*) or blank (‘’) entry refers to the standard set of fluid-in-place regions defined with the keyword \texttt{FIPNUM}.
   Additional families of fluid-in-place regions may optionally be defined, using the keyword \texttt{FIP}. The family name is string of up to 5 characters that makes up the second part of the keyword name (characters 4 - 8). Each family can divide up the field in different ways. Thus, for example, the standard \texttt{FIPNUM} family may divide the field into layers, and an additional family can be defined to divide the field into areal sectors.

4. The quantity to which the triggering condition applies.
   This must be one of:
   \begin{itemize}
   \item \texttt{RPR} Region pressure
   \item \texttt{ROSAT} Region oil saturation average value
   \item \texttt{RWSAT} Region water saturation average value
   \item \texttt{RGSAT} Region gas saturation average value
   \item \texttt{ROIP} Region oil in place
   \item \texttt{RWIP} Region water in place
   \item \texttt{RGIP} Region gas in place (in liquid and gas phases)
   \end{itemize}
   \textit{ECLIPSE 100 only}
   \begin{itemize}
   \item \texttt{RGIPG} Region gas in place (in gas phase only)
   \item \texttt{RUxxx} User defined region quantity
   \end{itemize}

5. The operator for the triggering condition.
   This must be one of:
   \begin{itemize}
   \item ‘>’ Greater than
   \item ‘<’ Less than
   \end{itemize}

6. The value of the triggering condition.
   The units depend upon the quantity selected in item 4.

7. Number of times this action can be triggered.
   The action is performed once at the end of each timestep while the triggering condition is satisfied, until it has been performed the specified number of times. A number of 10,000 or greater is interpreted as ‘infinity’.
   DEFAULT: 1

8. Increment to the triggering condition.
   After each time the action is performed, the specified increment is added to the triggering condition value (initialized in item 6). The increment may be negative, if desired.
DEFAULT: 0.0
End the data record with a slash (/)

CAUTION: It is possible to nest actions by including the definition of the inner action within the keyword set of the outer action (that is before its ENDACTIO keyword). This offers great flexibility, but care should be exercised and the consequences of the nested actions should be thought through. For example: suppose that in the ACTDIMS keyword we have set MAXACT=1 and have then defined an action ACT1. Suppose also that as part of this action we have defined another action, ACT2. ECLIPSE only issues an error when ACT1 is actually implemented which may, of course, be well into the run. Alternatively we may call the inner action by the same name as the outer action, ACT1. This is perfectly allowable, but it must be remembered that each time an action of a given name is defined it overwrites the properties that the action may previously have had. Suppose that we have an outer action that can be repeated a number of times. Within this action is an inner one that can also be repeated and that also increments its triggering condition each time. No matter how many times the inner action has been called when the outer action is invoked again the inner action is redefined to its original specification.

Note: Full checking of the keywords entered with an action keyword cannot be performed until the action is actually implemented, since it is possible to use wildcards in many of the keywords. However, before a simulation run is attempted the data can be run in NOSIM mode. This then does whatever checks it can on the keywords within any action keyword blocks that have been defined. All the most common types of 'grammatical' and typographical errors will be found, but errors resulting from wildcards feeding inappropriate well/group names to keywords are not found.

Example

This shows two action keyword sets. The first action (named ACT6) opens water injection well WINJNEW when the average pressure in the field (region 0) falls below 4000. The second action (named ACT7) reduces the oil rate target of well PROD3 by 20% whenever the average gas saturation in region 3 of the family WELRG exceeds 0.05.

```
ACTIONR
ACT6 0 1* RPR < 4000.0 /
WELOPEN
WINJNEW OPEN /
/
ENDACTIO
ACTIONR
ACT7 3 WELRG RGSAT > 0.05 10000 /
WTMULT
PROD3 ORAT 0.8 /
/
ENDACTIO
```
ACTIONS

Initiates a set of keywords to be processed when a well segment satisfies a specified condition

The ACTIONS keyword is one of a group of keywords that trigger actions in the simulation when certain conditions are met. While the ACTIONS keyword will work as documented in simulation datasets, it is recommended that you use the ACTIONX keyword for new or updated datasets as it offers the greatest functionality and flexibility.

The ACTIONS keyword marks the start of a set of SCHEDULE section keywords that are to be stored for later processing when a nominated segment of a multisegment well (see "Multisegment Wells" in the ECLIPSE Technical Description) satisfies a specified condition. The set of keywords may be indented with one or more blank spaces for clarity and must be terminated with the ENDACTIO keyword. The keywords between the ACTIONS and ENDACTIO keywords are processed at the end of the timestep when the condition defined in the ACTIONS keyword is satisfied by the well segment. You can choose whether the action is to be performed only once, or repeatedly each timestep while the triggering condition is satisfied.

If a well keyword (for example WELTARG) is placed between ACTIONS and ENDACTIO, instead of entering the name of a specific multisegment well in the keyword you may enter a question mark. ECLIPSE then applies the keyword data to whichever multisegment well (or wells) triggered the action. For example, an action can be placed on a set of multisegment wells ('PROD*' in item 2 of ACTIONS) to change their rate limit when a segment’s water cut exceeds a certain value. The keyword WELTARG would be entered between ACTIONS and ENDACTIO, specifying the new rate limit and applying it to the well named '?' in item 1.

Note: Some keywords cannot have the well name entered as a question mark to be inherited from an ACTIONS keyword. These exceptions are those keywords where name roots are not allowed for the well name. These keywords may still be used within an action if the well name is explicitly defined. However, well names can be inherited in the WELSPECS / WELSPECL keyword for ECLIPSE 100.

ECLIPSE 100

Practically any SCHEDULE section keyword may be placed between ACTIONS and ENDACTIO, with the exception of timestepping keywords (for example TIME, TSTEP, DATES and GASYEAR), the global keywords and the INCLUDE keyword. The RPTRST keyword can be used but any changes to the frequency of output will not the implemented until the report step following the satisfaction of the triggering condition.

ECLIPSE 300

Any keyword specifying data for wells, groups or separators may be placed between ACTIONS and ENDACTIO. The RPTRST keyword can also be used but any changes to the frequency of output will not be implemented until the report step following the satisfaction of the triggering condition.

The amount of storage space reserved for these keywords is controlled by the RUNSPEC section keyword ACTDIMS. The number of lines between an ACTIONS keyword and its corresponding ENDACTIO keyword (including the ENDACTIO keyword itself) is limited to MNLPAC in ACTDIMS item 2. The length of each line is limited to NCPLIN in ACTDIMS item 3.

Any number of action keyword sets may be declared, each with its own triggering condition; the maximum number being limited to MAXACT in ACTDIMS item 1. Each action keyword set must be bracketed by a pair of ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS) and ENDACTIO keywords, and is distinguished by its action name defined in item 1. This name must be unique across all the action keyword family. If an ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS) keyword is reentered with the same action name as a previously declared action, the previously declared action is deleted and replaced with the new action.

Keywords

266
The ACTIONS keyword contains the following items of data, terminated with a slash (/):

1. The action name.
   A name of up to 8 characters, to distinguish this action from any others.
   If an action with this name has already been declared in any of the action keyword family ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS), it will be overwritten with the action data specified below.

2. Well name (or name root) of the multisegment well(s) that triggers this action.
   A well name root, enclosed in quotes and ending with an asterisk (*), can be used to make this action’s triggering condition apply to more than one well, although the segment number that is monitored must be the same in each well. The action will then be performed if one or more multisegment wells having this name root satisfy the triggering condition in the nominated segment. Wells that conform to the name root but are not multisegment wells is ignored. Note that the action will be performed at most once per timestep, even if more than one well satisfies the triggering condition at the end of the timestep.

3. The number of the well segment whose quantity may trigger the action.
   An integer between 1 and NSEGMX (set in the RUNSPEC section keyword WSEGDIMS).

4. The quantity to which the triggering condition applies.
   This must be one of:
   - SOFR  Segment oil flow rate
   - SWFR  Segment water flow rate
   - SGFR  Segment gas flow rate
   - SGOR  Segment gas oil ratio
   - SOGR  Segment oil gas ratio
   - SWCT  Segment water cut
   - SWGR  Segment water gas ratio
   - SPR   Segment pressure
   - SPRD  Segment pressure drop
   - SOHF  Segment oil holdup fraction
   - SWHF  Segment water holdup fraction
   - SGHF  Segment gas holdup fraction
   - SUxxxxxx  User defined segment quantity

5. The operator for the triggering condition.
   This must be one of:
'>' Greater than
'<' Less than

6. The value of the triggering condition.
   The units depend upon the quantity selected in item 4.

7. Number of times this action can be triggered.
   The action is performed once at the end of each timestep while the segment set in item 3 of any
   multisegment well named in item 2 satisfies the triggering condition, until it has been performed the
   specified number of times. A number of 10,000 or greater is interpreted as ‘infinity’.
   DEFAULT: 1

8. Increment to the triggering condition.
   After each time the action is performed, the specified increment will be added to the triggering
   condition value (initialized in item 6). The increment may be negative, if desired.
   DEFAULT: 0.0

End the data record with a slash (/).

**CAUTION:** It is possible to nest actions by including the definition of the inner action within the keyword
set of the outer action (that is before its **ENDACTIO** keyword). This offers great flexibility, but care should
be exercised and the consequences of the nested actions should be thought through. For example: suppose
that in the **ACTDIMS** keyword we have set MAXACT=1 and have then defined an action **ACT1**. Suppose
also that as part of this action we have defined another action, **ACT2**. ECLIPSE only issues an error when
**ACT1** is actually implemented which may, of course, be well into the run. Alternatively we may call the
inner action by the same name as the outer action, **ACT1**. This is perfectly allowable, but it must be
remembered that each time an action of a given name is defined it overwrites the properties that the action
may previously have had. Suppose that we have an outer action that can be repeated a number of times.
Within this action is an inner one that can also be repeated and that also increments its triggering condition
each time. No matter how many times the inner action has been called when the outer action is invoked
again the inner action is redefined to its original specification.

**Note:** Full checking of the keywords entered with an action keyword cannot be performed until the action
is actually implemented, since it is possible to use wild cards in many of the keywords. However, before a
simulation run is attempted the data can be run in **NOSIM** mode. This then does whatever checks it can on
the keywords within any action keyword blocks that have been defined. All the most common types of
‘grammatical’ and typographical errors will be found, but errors resulting from wild cards feeding
inappropriate well/group names to keywords are not found.

**Example**

This shows two action keyword sets. The first action (named **ACT5**) applies to segment 5 of all
multisegment wells whose names begin with PR, and will reduce their liquid rate target by 20% whenever
the segment pressure falls below the value of 3500. The second action (named **ACT6**) opens well PR-A5
when the oil flow rate in segment 7 of any of the multisegment wells whose names begin with PR-A first
falls below 1,000.
ACTIONS
  ACT5 'PR**' 5 SPR < 3500 10000 /
  WTMULT
  '?' LRAT 0.8 /
  /
  ENDACTIO
ACTIONS
  ACT6 'PR-A**' 7 SOFR < 1000.0 /
  WELOPEN
  PR-A5 OPEN /
  /
  ENDACTIO
**ACTIONW**  Initiates a set of keywords to be processed when a well satisfies a specified condition

The ACTIONW keyword is one of a group of keywords that trigger actions in the simulation when certain conditions are met. While the ACTIONW keyword will work as documented in simulation datasets, it is recommended that you use the ACTIONX keyword for new or updated datasets as it offers the greatest functionality and flexibility.

The ACTIONW keyword marks the start of a set of SCHEDULE section keywords that are to be stored for later processing when a nominated well satisfies a specified condition. The set of keywords may be indented with one or more blank spaces for clarity and must be terminated with the ENDACTIO keyword. The keywords between the ACTIONW and ENDACTIO keywords will be processed at the end of the timestep when the condition defined in the ACTIONW keyword is satisfied by a well named in item 2. You can choose whether the action is to be performed only once, or repeatedly each timestep while the triggering condition is satisfied.

If a well keyword (for example WELTARG) is placed between ACTIONW and ENDACTIO, instead of entering the name of a specific well in the keyword you may enter a question mark. ECLIPSE then applies the keyword data to whichever well (or wells) triggered the action. For example, an action can be placed on a set of wells ('PROD*' in item 2 of ACTIONW) to change a well's rate limit when its water cut exceeds a certain value. The keyword WELTARG would be entered between ACTIONW and ENDACTIO, specifying the new rate limit and applying it to the well named '?' in item 1.

**Note:** Some keywords cannot have the well name entered as a question mark to be inherited from an ACTIONW keyword. These exceptions are those keywords where name roots are not allowed for the well name. These keywords may still be used within an action if the well name is explicitly defined. However, well names can be inherited in the WELSPECS / WELSPECL keyword for ECLIPSE 100.

Practically any SCHEDULE section keyword may be placed between ACTIONW and ENDACTIO, with the exception of time-stepping keywords (for example TIME, TSTEP, DATES and GASYEAR), the global keywords and the INCLUDE keyword. The RPTRST keyword can be used but any changes to the frequency of output will not the implemented until the report step following the satisfaction of the triggering condition.

Any keyword specifying data for wells, groups or separators may be placed between ACTIONW and ENDACTIO. The RPTRST keyword can also be used but any changes to the frequency of output will not be implemented until the report step following the satisfaction of the triggering condition.

The amount of storage space reserved for these keywords is controlled by the RUNSPEC section keyword ACTDIMS. The number of lines between an ACTIONW keyword and its corresponding ENDACTIO keyword (including the ENDACTIO keyword itself) is limited to MNLPAC in ACTDIMS item 2. The length of each line is limited to NCPLIN in ACTDIMS item 3.

Any number of action keyword sets may be declared, each with its own triggering condition; the maximum number being limited to MAXACT in ACTDIMS item 1. Each action keyword set must be bracketed by a pair of ACTION (ACTIONG, ACTIONR, ACTIONS, ACTIONW) and ENDACTIO keywords, and is distinguished by its action name defined in item 1. This name must be unique across all the action keyword family. If an ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS) keyword is re-entered with the same action name as a previously declared action, the previously declared action is deleted and replaced with the new action.

The ACTIONW keyword contains the following items of data, terminated with a slash (/):

---

**Keywords**

---

270
1. The action name.
   
   A name of up to 8 characters, to distinguish this action from any others.
   
   If an action with this name has already been declared in any of the action keyword family ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS), it is overwritten with the action data specified below.

2. Well name (or name root) of the well(s) that triggers this action.
   
   A well name root, enclosed in quotes and ending with an asterisk (*), can be used to make this action's triggering condition apply to more than one well. The action is then performed if one or more wells having this name root satisfy the triggering condition. Note that the action will be performed at most once per timestep, even if more than one well satisfies the triggering condition at the end of the timestep.

3. The quantity to which the triggering condition applies.
   
   This must be one of:

   **ECLIPSE 300 only**

   - **WWGPR** Well wet gas production rate

   Here are the other options:

   - **WOPR** Well oil production rate
   - **WOPT** Well oil production cumulative total
   - **WOIR** Well oil injection rate
   - **WOIT** Well oil injection cumulative total
   - **WWPR** Well water production rate
   - **WWPT** Well water production cumulative total
   - **WWIR** Well water injection rate
   - **WWIT** Well water injection cumulative total
   - **WGPR** Well gas production rate
   - **WGPT** Well gas production cumulative total
   - **WGIR** Well gas injection rate
   - **WGIT** Well gas injection cumulative total
   - **WLPR** Well liquid production rate
   - **WLPT** Well liquid production cumulative total
   - **WVPR** Well voidage production rate
   - **WVPT** Well voidage production cumulative total
   - **WVIR** Well voidage injection rate
   - **WVIT** Well voidage injection cumulative total
### Keywords

#### ECLIPSE 300 only
- `WWGPT`  [Well wet gas production cumulative total](#)
- `WWGIR`  [Well wet gas injection rate](#)
- `WWGIT`  [Well wet gas injection cumulative total](#)
- `WGOR`  [Well gas oil ratio](#)
- `WWCT`  [Well water cut](#)
- `WWGR`  [Well water gas ratio](#)
- `WGLR`  [Well gas liquid ratio](#)
- `WBHP`  [Well bottom hole pressure](#)
- `WTHP`  [Well tubing head pressure](#)

#### ECLIPSE 100 only
- `WBGLR`  [Well gas liquid ratio at bottom hole conditions](#)
- `WCPR`  [Well polymer production rate](#)
- `WCPC`  [Well polymer production concentration](#)
- `WSPR`  [Well salt production rate](#)
- `WSPC`  [Well salt production concentration](#)
- `WTFR`  [Well tracer production rate (tracer name needs to be added)](#)
- `WTPC`  [Well tracer production concentration (tracer name needs to be added)](#)

#### ECLIPSE 300 only
- `WNLPR`  [Well component molar rates in NGL phase](#)
- `WCNMR`  [Well NGL Production rate](#)
- `WUxxxxxx`  [User defined well quantity](#)

4. The operator for the triggering condition. This must be one of:
   - ‘>’ Greater than
   - ‘<’ Less than

5. The value of the triggering condition.
The units depend upon the quantity selected in item 3.

6. Number of times this action can be triggered.

The action will be performed once at the end of each timestep while any well named in item 2 satisfies the triggering condition, until it has been performed the specified number of times. A number of 10,000 or greater is interpreted as 'infinity'.

DEFAULT: 1

7. Increment to the triggering condition.

After each time the action is performed, the specified increment is added to the triggering condition value (initialized in item 5). The increment may be negative, if desired.

DEFAULT: 0.0

End the data record with a slash (/).

CAUTION: It is possible to nest actions by including the definition of the inner action within the keyword set of the outer action (that is before its ENDACTIO keyword). This offers great flexibility, but care should be exercised and the consequences of the nested actions should be thought through. For example: suppose that in the ACTDIMS keyword we have set MAXACT=1 and have then defined an action ACT1. Suppose also that as part of this action we have defined another action, ACT2. ECLIPSE only issues an error when ACT1 is actually implemented which may, of course, be well into the run. Alternatively we may call the inner action by the same name as the outer action, ACT1. This is perfectly allowable, but it must be remembered that each time an action of a given name is defined it overwrites the properties that the action may previously have had. Suppose that we have an outer action that can be repeated a number of times. Within this action is an inner one that can also be repeated and that also increments its triggering condition each time. No matter how many times the inner action has been called when the outer action is invoked again the inner action is redefined to its original specification.

Note: Full checking of the keywords entered with an action keyword cannot be performed until the action is actually implemented, since it is possible to use wild cards in many of the keywords. However, before a simulation run is attempted the data can be run in NOSIM mode. This then does whatever checks it can on the keywords within any action keyword blocks that have been defined. All the most common types of 'grammatical' and typographical errors will be found, but errors resulting from wild cards feeding inappropriate well/group names to keywords are not found.

Example

This shows two action keyword sets. The first action (named ACT5) applies to all wells whose names begin with PR, and reduces their liquid rate target by 20% whenever their BHP falls below the value of 3500. The second action (named ACT6) opens well PR-A5 when the oil production rate of any of the wells whose names begin with PR-A first falls below 1,000.

```plaintext
ACTIONW
  ACT5 'PR*' WBHP < 3500 10000 /
  WTMULT
    '?' LRAT 0.8 /
  /
ENDACTIO
ACTIONW
  ACT6 'PR-A*' WOPR < 1000.0 /
  WELOPEN
    PR-A5 OPEN /
```

Keywords
ENDACTIO
**ACTIONX**

**Initiates a set of keywords to be processed when a set of conditions are satisfied**

The ACTIONX keyword is a more flexible version of the ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS) type keywords. The keyword marks the start of a set of SCHEDULE section keywords that are to be stored for later processing when a set of conditions specified in this keyword are met. The set of keywords may be indented with one or more blank spaces for clarity and must be terminated with the ENDACTIO keyword. The keywords between the ACTIONX and ENDACTIO keywords will be processed at the end of the timestep when the set of conditions defined in the ACTIONX keyword is satisfied. You can choose whether the action is to be performed only once, or repeatedly each timestep while the triggering condition set is satisfied.

In ECLIPSE 100, practically any SCHEDULE section keyword may be placed between ACTIONX and ENDACTIO, with the exception of timestepping keywords (for example TIME, TSTEP, DATES and GASYEAR), the global keywords and the INCLUDE keyword. The RPTRST keyword can be used but any changes to the frequency of output will not be implemented until the report step following the satisfaction of the triggering condition.

Practically any SCHEDULE section keyword may be placed between ACTIONX and ENDACTIO, with the exception of timestepping keywords (for example TIME, TSTEP, DATES and GASYEAR), the global keywords and the INCLUDE keyword, and SIMULATE, NOSIM, SKIPREST, PSEUPRES and PICOND. The RPTRST keyword can be used but any changes to the frequency of output will not be implemented until the report step following the satisfaction of the triggering condition.

In the ACTIONG (ACTIONR, ACTIONW, ACTIONS) keywords the single triggering condition compares the current value of a quantity against a constant value defined in the keyword. For example, in ACTIONW the condition might be that a well’s GOR is greater than 3.0. In addition to this, the ACTIONX keyword also allows a comparison against the current value of another quantity. For example, a condition might be that a well’s GOR is greater than the current value of its controlling group’s GOR.

Multiple triggering conditions may be combined in the ACTIONX keyword using Boolean AND and OR operators to form the condition set. Conditions on quantities associated with well connections, well segments, wells, groups and regions may thus be combined in a single ACTIONX keyword.

Conditions may be set on the combined properties of a static (see WLIST keyword) or dynamic (see WLISTDYN keyword, ECLIPSE 300 only) list of wells. Grouping wells together using lists for the purposes of defining controlling conditions in an ACTIONX keyword offers more flexibility than conventional well groups. For example, a well in ECLIPSE 300 may belong to more than one list (see also WELLDIMS keyword item 11).

Conditions may be set on the day, month and year that the simulation has reached. Such a condition may be used to model the seasonal availability of workover and drilling rigs (see "Example 2"), or prevent the ACTION being triggered before a specific date (see "Example 4").

The amount of storage space reserved for these keywords is controlled by the RUNSPEC section keyword ACTDIMS. The number of lines between an ACTIONX keyword and its corresponding ENDACTIO keyword (including the ENDACTIO keyword itself) is limited to MNLPAC in ACTDIMS item 2. The length of each line is limited to NCPLIN in ACTDIMS item 3. The number of triggering conditions in an ACTIONX keyword is limited by item 4 in ACTDIMS.

Any number of action keyword sets may be declared, each with its own triggering condition(s); the maximum number being limited to MAXACT in ACTDIMS item 1. Each action keyword set must be bracketed by a pair of ACTIONG, ACTIONR, ACTIONW, ACTIONS and ENDACTIO keywords, and is
distinguished by its action name defined in item 1. This name must be unique across all the action keyword family. If an ACTIONG, ACTIONR, ACTIONW, ACTIONS keyword is reentered with the same action name as a previously declared action, the previously declared action is deleted and replaced with the new action.

The keyword is followed by a series of records each terminated by a slash (/) with the keyword terminated by a blank record containing only a slash (/):

The data items in each record are:

**Record 1**

1. The action name.
   - A name of up to 8 characters, to distinguish this action from any others.
   - If an action with this name has already been declared in any of the action keyword family ACTION (ACTIONG, ACTIONR, ACTIONS, ACTIONWACTIONX), it is overwritten with the action data specified below.

2. Number of times this action can be triggered.
   - The action is performed once at the end of each timestep while the set of conditions is satisfied, until it has been performed the specified number of times.
   - DEFAULT: 1

3. Minimum time interval between action triggers.
   - If this item is set, the action will only be performed if the set of conditions have been satisfied and the simulation has advanced beyond the specified time interval since the last time this action was triggered.
   - UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
   - DEFAULT: 0.0

**Each subsequent record**

Each of these records defines a condition on a region, group or well quantity.

A condition consists of the following elements:

1. A left hand quantity.
   - The quantity types and the items needed to define them, are described below.
   - If a well or group type quantity contains a root name followed by an asterisk (*), then this condition will be evaluated for more than one well or group. The condition will be deemed true if it is satisfied by at least one well or group. Similarly, in ECLIPSE 300, if a well connection quantity contains defaulted I, J or K locations, then the condition will be evaluated for more than one connection and deemed true if it is satisfied by at least one connection.

2. The operator for the triggering condition.
   - This must be one of:
     - '>' or '.GT.'  Greater than
     - '<' or '.LT.'  Less than

---

*Keywords*
'\textgreater{}' or '.GE.' Greater than or equals
'\textless{}=' or '.LE.' Less than or equals
'=' or '.EQ.' Equals
'\neq' or '.NE.' Not equals

3. A right hand quantity
   The quantity types and the items needed to define them, are described below.
   The use of well or group root names or defaulted I, J or K connection locations is not permitted in right hand quantities.

4. The Boolean operator that links this condition with those in subsequent records.
   This must be one of:
   \textbf{AND} applies a logical 'and' to this condition with the following
   \textbf{OR} applies a logical 'or' to this condition with the following

   If there are no subsequent conditions, leave this item defaulted.

End the data record with a slash (/) and end the keyword with a blank record containing only a slash (/).
Brackets may be placed before the left hand quantity and after the right quantity of a subsequent condition to allow greater flexibility in how conditions may be logically combined with the \textbf{AND} and \textbf{OR} boolean operators (see "Example 2"). Note that nesting brackets is not permitted. In the absence of brackets conditions are evaluated left to right.

The \textbf{"ALWAYS"} condition
This is an exception to the above format. This condition has no operators and no quantities; it consists of the single word \textbf{"ALWAYS"} (including the double quotes). The record and keyword are terminated as for all other conditions.

\textbf{"ALWAYS"} cannot be linked with conditions in subsequent records, as it has the effect that the keywords will be processed at the end of every timestep unconditionally. It is particularly intended for use with the \textbf{DYNAMICR} keyword, but is not restricted to it.

\textbf{Quantity types and definitions}

\textbf{Constant quantity}
This is defined by the following items:
1. Triggering condition value.
2. Increment to the triggering condition value defined in item 1.

   After each time the action is performed, the specified increment is added to the triggering condition value. The increment may be negative, if desired.
   DEFAULT: 0.0
**Time quantity**

1. This is defined by a single keyword. The following keywords can be used in both ECLIPSE 100 and ECLIPSE 300:
   
   **DAY**  Current simulation day of the month
   
   **MNTH** Current simulation month
   
   **YEAR** Current simulation year

   These quantities will be compared against the nearest integer values of a constant quantity on the other side of the operator in a condition. The **MNTH** keyword may also be compared against three letter abbreviations for calendar months (JAN, FEB ... DEC). See “Example 3” and “Example 4” for usage of this quantity type.

**Region quantity**

This is defined by the following items:

1. Region keyword. The following keywords can be used in both ECLIPSE 100 and ECLIPSE 300 unless other stated.
   
   This must be one of:

   **RPR** Region pressure
   
   **ROSAT** Region oil saturation average value
   
   **RWSAT** Region water saturation average value
   
   **RGSAT** Region gas saturation average value
   
   **ROIP** Region oil in place
   
   **RWIP** Region water in place
   
   **RGIP** Region gas in place (in liquid and gas phases)
   
   **RGIGP** Region gas in place (in gas phase only). This keyword can be used in ECLIPSE 100 only.
   
   **RUxxx** User defined region quantity

2. Fluid-in-place region number.
   
   This should be an integer between 0 and NTFIP (see keyword **REGDIMS** in the RUNSPEC section). Region 0 refers to the whole field.
   
   Fluid-in-place regions are defined with the keyword **FIPNUM** in the REGIONS section.

3. The fluid-in-place region family, to which the specified region belongs.
   
   The family name should be up to 5 characters long.

   A default (1*) or blank (‘’) entry refers to the standard set of fluid-in-place regions defined with the keyword **FIPNUM**.

   Additional families of fluid-in-place regions may optionally be defined, using the keyword **FIP**. The family name is string of up to 5 characters that makes up the second part of the keyword name.
Each family can divide up the field in different ways. Thus, for example, the standard FIPNUM family may divide the field into layers, and an additional family can be defined to divide the field into areal sectors.

**Field quantity**

This is defined by the following item:

1. Field keyword. The allowed mnemonics are all field keyword listed in the `SUMMARY` section.

**Keywords with a component index**

*ECLIPSE 300*

Keywords that provide information at a component level must have the component number appended to the keyword with an underscore, see Examples 5 and 6.

**Group or well list quantity**

This is defined by the following items:

1. Group keyword. The allowed mnemonics are all group keywords listed in the `SUMMARY` section.

**Keywords with a component index**

*ECLIPSE 300*

Keywords that provide information at a component level must have the component number appended to the keyword with an underscore, see Examples 5 and 6.

2. Group name (or name root, enclosed in quotes and ending with an asterisk - these are not permitted in a right hand quantity). A well list name may be specified for this item. The quantity defined in item 1, will then be calculated for the well grouping currently defined by this list.

**Well quantity**

This is defined by the following items:

1. Well keyword. The allowed keywords are all well keywords listed in the `SUMMARY` section.

2. Well name (or name root, enclosed in quotes and ending with an asterisk - these are not permitted in a right hand quantity). The well name may be defaulted by using the character `*` (this is not permitted in a right hand quantity).

**Keywords with a component index**

*ECLIPSE 300*

Keywords that provide information at a component level must have the component number appended to the keyword with an underscore, see Examples 5 and 6.

**Keywords with a lumped completion index**

Well keywords that need a lumped completion index must have this index appended to the keyword with underscores (two if the index is less than 10, one if it is less than 100, none otherwise).

**Well segment quantity**

This is defined by the following items:

1. Well segment keyword. The allowed keywords are all those segment keywords listed in the `SUMMARY` section.

2. Multisegment well name (or name root, enclosed in quotes and ending with an asterisk - these are not permitted in a right hand quantity).
If a well name root is specified, then wells that conform to the name root but are not multisegment wells are ignored.

3. Well segment number.
   An integer between 1 and NSEGMAX (set in the RUNSPEC section keyword WSEGDIMS)

Well connection quantity

1. Well connection keyword. The allowed keywords are all those connection keywords listed in the SUMMARY section.

2. Well name (or name root, enclosed in quotes and ending with an asterisk - these are not permitted in a right hand quantity). The well name may be defaulted by using the character ‘*’ (this is not permitted in a right hand quantity).

3. I - location of connecting grid block(s)
   DEFAULT: Negative (allows any I-location)

4. J - location of connecting grid block(s)
   DEFAULT: Negative (allows any J-location)

5. K - location of connecting grid block(s)
   DEFAULT: Negative (allows any K-location)

If any of items 3, 4, or 5 are set negative, then a left hand quantity will be evaluated for more than one connection in the well. These connections are those with I,J,K locations matching the specified location indices in items 3, 4, or 5. If the well name is defaulted using the character ‘*’, the left hand quantity will be evaluated for all wells in the global model. Note that it is not permitted to default connection locations or well name if this is a right hand quantity.

Well LGR connection quantity

This quantity refers to connections in a local grid refinement. It is defined by the following items:

1. Well connection keywords. The allowed keywords are all those connection keywords listed in the SUMMARY section. These keywords should be prefixed by an ‘L’ to indicate an LGR connection quantity (as is the case for summary mnemonics)

2. Well name (or name root, enclosed in quotes and ending with an asterisk - these are not permitted in a right hand quantity). The well name may be defaulted by using the character ‘*’ (this is not permitted in a right hand quantity).

3. Local grid name

4. I - location of connecting grid block(s) within the local grid
   DEFAULT: Negative (allows any I-location)

5. J - location of connecting grid block(s) within the local grid
   DEFAULT: Negative (allows any J-location)

6. K - location of connecting LGR grid block(s) within the local grid
   DEFAULT: Negative (allows any K-location)

If any of items 4, 5, or 6 are set negative, then a left hand quantity will be evaluated for more than one connection in the well. These connections are those with I,J,K locations matching the specified location
indices in items 4, 5, or 6 within the local grid. If the well name is defaulted using the character '*', the requested connection quantities are evaluated for all wells in the specified LGR. Note that it is not permitted to default connection locations or well name if this is a right hand quantity.

**Aquifer quantity**

This is defined by the following items:

1. Aquifer keyword. The allowed keywords are all aquifer keywords listed in the `SUMMARY` section.
2. Aquifer number

**Block quantity**

1. Block keyword. The allowed keywords are all block keywords listed in the `SUMMARY` section.
2. I - location of grid block
3. J - location of grid block
4. K - location of grid block

**Notes**

- The order of the items in the quantities is similar to the way summary mnemonics are defined, that is the mnemonic is the first item and it is then followed by a well or group name for example. This is to be contrasted with the order of these items in the `ACTION` (`ACTIONG`, `ACTIONR`, `ACTIONW`, `ACTIONS`) type keywords.

- A condition consisting of constant quantities on the left and right hand sides is not permitted. A condition may, however, consist of a constant quantity on the left hand side and a variable quantity on the right hand side.

- No checks are in place on the type of quantities that are compared in a condition. So for example, a condition on a region’s pressure being less than a connection’s water cut would be allowed.

- If sufficient conditions are met for the `ACTIONX` keyword to trigger the processing of its set of `SCHEDULE` section keywords, then details of those conditions that are true will be written to the print file.

- Combined rates, totals and ratios for all the wells in a well list may be written to the summary file using an appropriate group `SUMMARY` keyword followed by the well list name.

- If a well keyword (for example `WELTARG`) is placed between `ACTIONX` and `ENDACTIO`, instead of entering the name of a specific well in the keyword you may enter a question mark (?). ECLIPSE then applies the keyword data to whichever well (or wells) are involved in triggering the action. See "Examples".

If there is more than one condition in the action that concerns wells, different sets of wells may trigger different conditions. These set of triggering wells are combined logically in the same manner as the conditions themselves. The question mark in the well keywords will then apply to this combined set of wells. Note that if conditions are combined with logical AND operators, there may be no wells that trigger every condition and thus the well keywords will not have any effect, even though the overall action has been triggered.

- Full checking of the keywords entered with an action keyword cannot be performed until the action is actually implemented, since it is possible to use wild cards in many of the keywords. However, before a simulation run is attempted the data can be run in `NOSIM` mode. This then does whatever checks it
can on the keywords within any action keyword blocks that have been defined. All the most common
types of ‘grammatical’ and typographical errors will be found, but errors resulting from wild cards
feeding inappropriate well/group names to keywords are not found.

• An action may be triggered on a grid block quantity by defining a region to consist of one grid block
(keyword FIPNUM) and selecting a region quantity in a condition.

Examples

Example 1

The action named ACT1 reduces the liquid rate target by 20% of all wells starting with ‘PR’ whose GOR
exceeds that of the FIELD when the gas production rate of the FIELD exceeds 50000 Mscf/day. The action
will be repeated at the end of each timestep whilst the condition set is true.

```
ACTIONX
   ACT1 100000 /
   GGPR FIELD > 50000 AND /
   WGOR 'PR*' > GGOR FIELD /
   /
   WTMULT
   '?*' LRAT 0.8 /
   /
   ENDACTIO
```

Example 2

The action named ACT2 opens well PR-A5 when the combined water cut of all the wells in list *LIST1
exceeds 0.7 and when the water production rate of list *LIST1 exceeds the water production rate of either
list *LIST2 or *LIST3. Action ACT2 is performed once only.

```
ACTIONX
   ACT2 1 /
   GWCT '*LIST1' > 0.7   AND   /
   ( GWPR '*LIST1' > GWPR '*LIST2' OR    /
     GWPR '*LIST1' > GWPR '*LIST3' ) /
   /
   WELOPEN
   PR-A5 OPEN /
   /
   ENDACTIO
```

Example 3

The dynamic well list *HIGHWCT contains all wells that have a water cut exceeding 0.8. In any year
between March and October, the action WORKOVER will be triggered if the list *HIGHWCT contains more
than 4 wells This action will set an economic limit on the wells in the *HIGHWCT list such that they will be
worked over. The WORKOVER action also contains a nested action WECONOFF that is triggered when the
simulation advances past October. WECONOFF stops any further workovers in the winter by re-specifying
the WECON keyword with a high water cut limit.

```
ACTIONX
   WORKOVER 10000 /
```

Keywords
Example 4

The action named DELAY opens well PR-A5 when the simulation has advanced past the 1st June 2021 and the FIELD’s water cut exceeds 0.8.

Example 5

The action named ACT5 reduces the liquid rate target by 20% of all wells starting with ‘PR’ when the 2nd component molar rate in the gas phase of the well ‘PROD1’ exceeds 5th component molar rate in the gas phase of the well ‘PROD2’ and when the 3rd component molar rate in the gas phase of the group ‘G1’ exceeds 7th component molar rate in the gas phase of the group ‘G2’ or when the 1st component molar rate in the gas phase of the FIELD exceeds the 7th component molar rate in the gas phase of the FIELD. That is, the first two conditions must be true, or the third. The action will be repeated at the end of each timestep whilst the condition set is true.
Example 6

The action named ACT4 reduces the liquid rate target by 50% of all wells starting with ‘PR’ when the user defined quantity WUMW of the well ‘PROD1’ exceeds WUMW of the well ‘PROD2’ and when the user defined quantity GUMW of the group ‘G1’ exceeds GUMW of the group ‘G2’ or when the user defined quantity FUMW of the FIELD less than 1. The action will be repeated at the end of each timestep whilst the condition set is true.

```
Example 6

UDQ -- A COUPLE OF CONNECTION QUANTITES
DEFINE WUMW WCGMR_2/(MAX(WCGMR_5)+1) /
UNITS WUMW WELL/
DEFINE GUMW GCGMR_3/(GCGMR_5+GCGMR_1) /
UNITS GUMW GROUP/
DEFINE FUMW FCGMR_3/(FCGMR_5+FCGMR_7) /
UNITS FUMW FIELD/

ACTIONX
    ACT4 1000000 /
    WUMW 'PROD1' > WUMW'PROD2' AND /
    GUMW 'G1' > GUMW 'G2' OR /
    FUMW < 1 /
/
WTMULT
    '?' LRAT 0.5 /
/
ENDACTIO
```
ACTNUM  Active grid block identification

The keyword should be followed by one integer for every grid block in the current box.

A value of 1 indicates that the corresponding grid block is active, whilst a 0 indicates that it is inactive. If the ECLIPSE 300 Thermal option (keyword THERMAL in the RUNSPEC section) is requested, a value of 2 indicates that the grid block has rock volume only, and a value of 3 indicates that the grid block has pore volume only.

The data must be terminated by a slash (/).

It should be noted that this is only one of a number of ways of identifying inactive grid blocks. Other possibilities include setting the porosity (PORO), or net to gross thickness ratio (NTG), to zero in the GRID section.

Indeed, any method which results in zero pore volume will cause a grid block to be treated as inactive. For example, the MINPV keyword can be used to set a minimum threshold pore volume, so that any grid blocks having a pore volume smaller than this, will be treated as inactive.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 12*0). Note that spaces must not be inserted on either side of the asterisk.

ECLIPSE 300 For runs with the Thermal option and with Dual or Multi Porosity, the combination of available ACTNUM values can cause inconsistencies in how the data is used. A wholly inactive (ACTNUM=0) matrix or fracture cell will deactivate the corresponding fracture or matrix, but other values will affect only the selected porosity and leave the other porosities unaltered. However, the meaning of, for example, a rock-only fracture cell, or a fluid-only matrix cell, is ill-defined, and the user should take care in using these options. The issue is discussed further in “Dual porosity” in the ECLIPSE Technical Description.

ECLIPSE 300 For runs with the Thermal option and with Dual or Multi Porosity, also note that ACTNUM=3, that is, pore volume only, means that the porosity will be set to 1.0, regardless of the input porosity values. Some control over the fraction of cell volume which is pore volume is still available with this option by use of the net to gross values (NTG).

ECLIPSE 300 The ACTNUM values set using this keyword can be output by setting item 5 of the DEBUG3 keyword to a non-zero positive value. The active cell indicator information is output to both the GRID and EGRID files. Please refer to the description provided in the ECLIPSE File Formats Reference Manual. This array of information is further updated internally to provide a natural cell to active cell mapping (refer to additional output given by this DEBUG3 switch as well as the IOCNR argument for use with the RPTGRID and RPTGRIDL keywords).

Example

With NDIVIX=8, NDIVIY=6, NDIVIZ=3 in RUNSPEC keyword DIMENS:

```
ACTNUM
0 0 0 1 1 1 0 0
0 1 1 1 1 1 0 0
1 1 1 1 1 0 0 0
0 1 1 1 1 1 1 1
0 0 1 1 0 1 1 1
0 0 1 0 0 1 1 0
0 0 0 1 1 1 0 0
0 0 0 1 1 1 0 0
1 1 0 0 1 0 0 0
0 1 1 1 1 1 1 1
0 0 1 1 0 1 0 1
0 0 1 0 0 0 0 0
```

Keywords

285
ACTPARAM  Parameters for the ACTION facility

The data consists of the following items, specifying control parameters associated with the ACTION keyword family. The data must be terminated by a slash (/).

1. The (target) maximum percentage violation of an ACTIONX type condition.
   
   DEFAULT: 100.0% (Target not applied)

   **Note:** When using this item there is no guarantee that the condition will not be violated by more than the percentage set here; a timestep cannot be repeated just because a condition has been violated by more than the target percentage. If the values on both sides of a condition are smoothly varying (in time) then item 1 will usually perform well. If one side of the condition changes abruptly, for example a group rate may change discontinuously due to wells shutting in, then this item may not help.

2. The default fractional equality tolerance used in deciding whether quantities are equal in an ACTIONX condition that contains an equals type comparator.
   
   DEFAULT: 1.0E-4

See keywords ACTION, ACTIONG, ACTIONR, ACTIONW, ACTIONS, ACTIONX and ENDACTIO.

**Example**

This example shows that the condition will be violated by not more than 3 percentage set in ACTPARAM keyword. Note that the number of times this condition can be triggered must be greater than 2.

```
ACTPARAM
3.0 /
```

Keywords
**ADD**

**Adds a constant to the specified array in current box**

The keyword may be followed by any number of records, each of which is terminated by a slash (/). The data is terminated by a blank record, containing only a slash.

Each record consists of at least 2 items of data:

1. The name of the array to be modified
2. The constant to be added to the array specified by item 1

   The constant must be an integer if the array contains integer region numbers (for example, `FIPNUM`). The constant may be real if the array contains real data (for example, `TOPS` or `TRANX`). This item should not be defaulted.

   The following data items define the limits of a box within the grid over which the value is to be added.

   These items may be used to redefine the input box for this and subsequent operations within the current keyword. The values are used until reset or until the end of the keyword.

   If these items are not defined (a slash is inserted after item 2), they default to the values which were used for the previous operation within the current keyword.

   For the first operation in the keyword, the box defaults to the values set by the most recent `BOX` or `ENDBOX` keyword. If there is no preceding `BOX` or `ENDBOX` in the current section, the box is taken to include the entire reservoir.

3. First block to be modified on the X axis (IX1)
4. Last block to be modified on the X axis (IX2)
5. First block to be modified on the Y axis (JY1)
6. Last block to be modified on the Y axis (JY2)
7. First block to be modified on the Z axis (KZ1)
8. Last block to be modified on the Z axis (KZ2)

   The data must satisfy:

   \[1 \leq IX1 \leq IX2 \leq NDX\]

   \[1 \leq JY1 \leq JY2 \leq NDY\]

   \[1 \leq KZ1 \leq KZ2 \leq NDZ\]

   where NDX, NDY, and NDZ are the limits in the current BOX.

   If the `ADD` keyword is used in the `PROPS` section to modify the end-point scaling arrays, the addition is only made to the end points that have been previously defined. If a cell in the input box has not been assigned a scaled end point, `ADD` does not operate on this cell, and its end point still defaults to the table value.

   `TNUM` arrays can also be used with the `ADD` keyword.

---

**Keywords**

---

288
Legal arrays

GRID section arrays

<table>
<thead>
<tr>
<th>DX (DR)</th>
<th>DY (DTHETA)</th>
<th>DZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>PERMX (PERMR)</td>
<td>PERMY (PERMTHT)</td>
<td>PERMZ</td>
</tr>
<tr>
<td>MULTX (MULTR)</td>
<td>MLYT (MULTHT)</td>
<td>MULTZ</td>
</tr>
<tr>
<td>PORO</td>
<td>NTG</td>
<td>DZNET</td>
</tr>
<tr>
<td>TOPS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>DIFFMX (DIFFMR)</td>
<td>DIFFMY (DIFFMTHT)</td>
<td>DIFFMZ</td>
</tr>
<tr>
<td>FLUXNUM</td>
<td>MULTNUM</td>
<td>MPANUM</td>
</tr>
</tbody>
</table>

ECLIPSE 300 only

MIDS

EDIT section arrays

<table>
<thead>
<tr>
<th>PORV</th>
<th>DEPTH</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRANX (TRANR)</td>
<td>TRANY (TRANTHT)</td>
</tr>
<tr>
<td>DIFFX (DIFFR)</td>
<td>DIFFY (DIFFHTH)</td>
</tr>
</tbody>
</table>

ECLIPSE Thermal only

HEATTX (HEATTR) HEATTY (HEATTHT) HEATTZ (HEATTZ)

ECLIPSE Thermal only

ROCKV

PROPS section arrays

<table>
<thead>
<tr>
<th>SWL</th>
<th>SWCR</th>
<th>SWU</th>
</tr>
</thead>
<tbody>
<tr>
<td>SGL</td>
<td>SGC</td>
<td>SGU</td>
</tr>
<tr>
<td>KRW</td>
<td>KRO</td>
<td>KRG</td>
</tr>
<tr>
<td>PCG</td>
<td>PCW</td>
<td></td>
</tr>
</tbody>
</table>

ECLIPSE 300 only

PSORG PSORW PSGRW

ECLIPSE 300 only

PSGRO PSGW PSGRW

ECLIPSE 300 only

HSORG HSORW HSGRW

ECLIPSE 300 only

HSGRO HSWRO HSWRG

REGIONS section arrays

<table>
<thead>
<tr>
<th>SATNUM</th>
<th>PVTNUM</th>
<th>EQLNUM</th>
</tr>
</thead>
<tbody>
<tr>
<td>IMBNUM</td>
<td>FIPNUM</td>
<td>ENNUM</td>
</tr>
</tbody>
</table>
ROCKNUM  MISCNUM  WH2NUM
WH3NUM

ECLIPSE 300 only
EOSNUM

ECLIPSE 300 only
PSTNUM  ISTNUM  DSTNUM

ECLIPSE 300 only
SDROW  SDROG  SDRGW

ECLIPSE 300 only
SDRWO  SDRGO  SDRWG

**SOLUTION section arrays**

PRESSURE  SWAT  SGAS
RV  RS  TBLK

ECLIPSE 100 only
GI  OILAPI  SALT

ECLIPSE 100 only
GASCONC  SOLVCONC  SOLVFRAC

ECLIPSE 100 only
SSOL  SURF  SFOAM

ECLIPSE 100 only
SPOLY

ECLIPSE 300 only
SOIL  PBUB  PDEW

ECLIPSE 300 only
TEMP  TEMPI

**Notes**

- The above list is not exhaustive, but includes the most common examples.
- Keywords in brackets are for radial geometry runs.
- In the *REGIONS* section only integer values should be used.
- The added quantity may be negative, in order to subtract.
- The use of the *ADD* keyword assumes that a value has already been assigned to the array, either explicitly, or using the *EQUALS* keyword (except in the *EDIT* section, where the array has already been calculated after processing the *GRID* section).
- The *ADD*, *COPY*, *EQUALS* and *MULTIPLY* keywords are processed as they are read, so that repeated operations are possible.

See also the keywords *BOX*, *ENDBOX*, *EQUALS*, *MULTIPLY* and *COPY*.

**Item 2 should not be defaulted.** However, this facility is retained for back-compatibility. If this item is defaulted, this will be treated differently by ECLIPSE 100 and ECLIPSE 300. For ECLIPSE 100, the defaulted value is assumed to be undefined, whereas for ECLIPSE 300, the defaulted value is assumed to be zero. The subsequent processing will depend, therefore, both upon the specific keyword to which the operation is being applied and which simulator is being used.
Examples

Example 1

In the REGIONS section:

```
-------- ARRAY     CONSTANT      ----- BOX ----- 
ADD       
  SATNUM  3        1 11 1 19 2 2 /  
  PVTNUM  1        / defaults to last specified box 
  EQLNUM  1        / defaults to last specified box 
  FIPNUM  8        / defaults to last specified box 
/ 
```

Example 2

In the EDIT section:

```
-------- ARRAY     CONSTANT      ----- BOX ----- 
ADD       
  PORV   350.       0 1 11 1 19 2 2 /  
  DEPTH  1        / defaults to last specified box 
  TRANX  1        / defaults to last specified box 
/ 
```
ADDREG

**Adds a constant to the specified array in either a flux region, MULTNUM region or OPERNUM region**

ADDREG is similar to the ADD keyword, but uses either FLUXNUM regions, MULTNUM regions or OPERNUM regions, rather than boxes of cells.

For the FLUXNUM case, there is no requirement to use the Flux option.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated by a slash (/).

The records can be terminated early using the slash character (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

Each record consists of the following items of data:

1. The name of the array to be modified
2. The constant to be added to the array, specified by item 1
   The constant may be positive or negative, and, for the REGIONS section, should be integer.
   DEFAULT: 0.0
3. The flux region number, MULTNUM region or OPERNUM region number (with choice of region determined using item 4)
   The flux region number refers to the region defined using the FLUXNUM keyword in the GRID section. It is not necessary to use the Flux Boundary option.
   The MULTNUM region number refers to the region defined using the MULTNUM keyword in the GRID section.
   The OPERNUM region number refers to the region defined using the OPERNUM keyword in the GRID or REGIONS section.
4. Choice of region: MULTNUM (M), FLUXNUM (F) or OPERNUM (O).
   DEFAULT: M

The set of records is ended by a blank record, containing only a slash (/).

**Note:** The FLUXNUM, MULTNUM or OPERNUM keyword must be input before the ADDREG keyword.

---

**ECLIPSE 100**

This keyword may not be used to modify the PBUB or PDEW arrays.

**Note:** If item 4 is defaulted, then the MULTNUM keyword is used.

---

**ECLIPSE 300**

This keyword may be used in the SCHEDULE section. However, this is a specialist facility.
**Legal arrays**

**GRID section arrays**

- $DX$ (DR)  $DX$ (DTHETA)  $DZ$
- $PERMX$ (PERMR)  $PERMY$ (PERMTHT)  $PERMZ$
- $MULTX$ (MULTR)  $MULTY$ (MULTTHT)  $MULTZ$
- $PORO$  $NTG$  $DZNET$
- $TOPS$
- $DIFFMX$ (DIFFMR)  $DIFFMY$ (DIFFMTHT)  $DIFFMZ$

_ECLIPSE 300 only_

- $FLUXNUM$
- $MULTNUM$
- $MPFANUM$

_ECLIPSE 300 only_

- $MIDS$

**EDIT section arrays**

- $PORV$  $DEPTH$
- $TRANX$ (TRANR)  $TRANY$ (TRANTHT)  $TRANZ$
- $DIFFX$ (DIFFR)  $DIFFY$ (DIFFTHT)  $DIFFZ$

_ECLIPSE Thermal only_

- $HEATTX$ (HEATTR)  $HEATTY$ (HEATTHT)  $HEATTZ$

_ECLIPSE Thermal only_

- $ROCKV$

**PROPS section arrays**

- $SWL$  $SWCR$  $SWU$
- $SGL$  $SGCR$  $SGU$
- $KRW$  $KRO$  $KRG$
- $PCG$  $PCW$

_ECLIPSE 300 only_

- $PSORG$  $PSORW$  $PSGRW$

_ECLIPSE 300 only_

- $PSGRO$  $PSWRO$  $PSWRG$

_ECLIPSE 300 only_

- $HSORG$  $HSORW$  $HSGRW$

_ECLIPSE 300 only_

- $HSGRO$  $HSWRO$  $HSWRG$

**Note:** In the SCHEDULE section, regions (pre-defined by the FIPxxxx keyword in the REGIONS section) can be used in item 4.
REGIONs section arrays
SATNUM  PVTNUM  EQLNUM
IMBNUM  FIPNUM  ENDNUM
ROCKNUM MISCMUM  WH2NUM
WH3NUM
ECLIPSE 300 only
EOSNUM
ECLIPSE 300 only
PSTNUM  ISTNUM  DSTNUM
ECLIPSE 300 only
SDROW  SDROG  SDRGW
ECLIPSE 300 only
SDRWO  SDRGO  SDRWG

SOLUTION section arrays
PRESSURE  SWAT  SGAS
RV   RS  TBLK
ECLIPSE 100 only
GI  OILAPI  SALT
ECLIPSE 100 only
GASCONC  SOLVCONC  SOLVFRAC
ECLIPSE 100 only
SSOL  SPOLY  SURF
ECLIPSE 100 only
SFOAM
ECLIPSE 300 only
SOIL  PBUB  PDEW
ECLIPSE 300 only
TEMP

SCHEDULE section arrays (specialist facility)
ECLIPSE 300 only
PORO  PERMX  PERMY
ECLIPSE 300 only
PERM2  MULTX  MULTY
ECLIPSE 300 only
MULTZ  SIGMA(V)  MULTSIG(V)
ECLIPSE 300 only
MULSGGD(V)  DIFFMMF  WORK(1/2)

Notes
• The above list is not exhaustive, but includes the most common examples.
• Keywords in brackets are for radial geometry runs.
• In the REGIONs section only integer values should be used.
• The added quantity may be negative, in order to subtract.
• The use of the ADDREG keyword assumes that a value has already been assigned to the array.
• Each record must refer to the same region. It is not possible to choose a different region for each data record.

**ECLIPSE 100 only**

• The ADDREG keyword is not available for use with local grid systems.

See also keywords **EQUALREG, FLUXNUM, MULTNUM, OPerNUM, MULTIREG** and **COPYREG**.

**Additional notes**

**ECLIPSE 300 only**

Additional notes for the use of this keyword in the **SCHEDULE** section. Again, this is a specialist facility. In the **SCHEDULE** section, the effect of this keyword will be as follows:

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DIFFMMF</td>
<td>Modifies the matrix-fracture diffusivity in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as DIFFMMFS.</td>
</tr>
<tr>
<td>MULSGGD</td>
<td>Modifies the matrix-fracture coupling transmissibility for oil-gas gravity drainage in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as MULSGGDS.</td>
</tr>
<tr>
<td>MULSIG</td>
<td>Modifies the matrix-fracture coupling transmissibility in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as MULSIGS.</td>
</tr>
<tr>
<td>MULTX</td>
<td>Modifies transmissibility in the X/Y/Z-direction. If output is required for flexible restart in the model, the multiplier values will be output as MULTXS/MULTYS/MULTZS.</td>
</tr>
<tr>
<td>PERMX</td>
<td>Modifies permeability in the X/Y/Z-direction. Also approximates the corresponding transmissibility multiplier MULTX/Y/Z determined from the ratio of the new permeability value to the old one. If output is required for flexible restart in the model, the permeability values will be output as MODPERMX/Y/Z.</td>
</tr>
<tr>
<td>PORO</td>
<td>Modifies porosity. Modifies the corresponding pore volume by a multiplier MULTPV determined from the ratio of the new porosity value to the old one. If output is required for flexible restart in the model, the porosity values will be output as MODPORO, and the pore volume multipliers will be output as MULTPVPS.</td>
</tr>
<tr>
<td>SIGMA</td>
<td>Modifies the matrix-fracture coupling transmissibility in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as SIGMA.</td>
</tr>
<tr>
<td>WORK (1/2)</td>
<td>These are work arrays to be stored for subsequent use. If output is required for flexible restart in the model, these values will be output as WORK (1/2).</td>
</tr>
</tbody>
</table>

*Table 3.1: Additional notes for ADDREG in the SCHEDULE section*

**Examples**

**Example 1**

Where \( NX=4, \) \( NY=4, \) \( NZ=2 \):

```
FLUXNUM
1 1 1 1
1 1 1 1
1 1 1 1
1 1 1 1
```
Example 2

NX=6, NY=2, NZ=1:

```
MULTNUM
  1 1 2 2 2 2
  1 1 1 2 2 2 /
ADDREG
PORO  50.0  1 /
PERMY 25.0  2 /
/```

ADDREG
DZ     5.0    1    F / Add 5 feet to the top layer (region 1).
PERMX  100.0   3    F / Add 100 md to region 3.
ADDZCORN  Adds a constant to the corner point depth array

The keyword may be followed by any number of records, each of which is terminated by a slash (/). The data is terminated by a null record (that is a record with no data before the terminating slash). Each record consists of at least one item of data.

Items 2-7 are used to redefine the input box for the modification to the corner depths. The constant specified in item 1 is added to all the corners within this box, with the possible exception of the bottom corners of the bottom layer (KZ = KZ2), or the top corners of the top layer (KZ = KZ1), see item 12 below.

The modification is either continuous with the surrounding cells or discontinuous, depending on the settings of data items 8 to 11.

1. The constant to be added to the corner point (ZCORN) array. The constant may be positive or negative; a positive constant increases the depths of the cell corner points.

A record can also contain the following data items.

If items 2-7 are not defined (a slash is inserted after item 1), they default to the values which were used for the previous operation within the current keyword. For the first operation in the keyword, the box defaults to the values set by the most recent BOX or ENDBOX keyword. If there is no preceding BOX or ENDBOX in the current section, the box is taken to include the entire reservoir.

2. IX1 First block to be modified on the X axis.
3. IX2 Last block to be modified on the X axis.
4. JY1 First block to be modified on the Y axis.
5. JY2 Last block to be modified on the Y axis.
6. KZ1 First block to be modified on the Z axis.
7. KZ2 Last block to be modified on the Z axis.

The data must satisfy

\[
0 \leq IX1 \leq IX2 \leq NDIVIX \\
0 \leq JY1 \leq JY2 \leq NDIVIY \\
0 \leq KZ1 \leq KZ2 \leq NDIVIZ
\]

Where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS.

However, if only certain corners within a block are to be modified, any of IX1, IX2, JY1 or JY2 may be zero (consider the Examples).

Items 8-11 are used to specify whether the modification should be continuous with the surrounding cells.

8. IX1A = IX1 -1 or IX1
   DEFAULT: IX1 -1 if IX1 > 1
9. IX2A = IX2 +1 or IX2
   DEFAULT: IX2 +1 if IX2 < NDIVIX
10. JY1A = JY1 -1 or JY1
DEFAULT: JY1 -1 if JY1 >1

11. JY2A = JY2 +1 or JY2
   DEFAULT: JY2 +1 if JY2 < NDIVIY

   The data must satisfy

   0 ≤ IX1A ≤ NDIVIX 0 ≤ IX2A ≤ NDIVIX
   0 ≤ JY1A ≤ NDIVIY, 0 ≤ JY2A ≤ NDIVIY

   If, for example, IX2A = IX2, then blocks with I = IX2 +1 are not modified and a discontinuity (or
fault) is added to the surface. However, if IX2A = IX2 +1, then the corner points in blocks at I =
IX2 +1 adjacent to the blocks at I = IX2 have the constant added, so preserving a continuous surface.

   If the surface is discontinuous between I = IX2 and I = IX2 +1 prior to the action of the ADDZCORN
keyword, then no action is taken with the blocks at I = IX2 +1.

   By default items 8-11 are set to give a continuous surface.

12. Action required:
   ALL Modify both top and bottom corner points of all cells.
   TOP Modify both top and bottom corner points of all cells, except for the bottom layer, in which
the top cells only are modified.
   BOTTOM Modify both top and bottom corner points of all cells, except for the top layer, in which the
bottom corners, only are modified.

   The word may be abbreviated as only the first character is significant.

   DEFAULT: ALL

   See also the BOX, ENDBOX, ADD and EQLZCORN keywords.

Modifying selected corner point depths within a grid block

It is possible to modify selected corner depths within a grid block by specifying a zero in either the
IX1, IX2 or JY1, JY2 box limits (or both). In this case only the corner depth on the appropriate side of the
cell will be modified.

If data items 8-11 are defaulted or set to a box outside the box defined in data items 2-7, then the adjacent
cell depths are modified to give a continuous surface. Note that if the surface was discontinuous prior to the
ADDZCORN keyword, the adjacent cells are not modified. See Example 2.

Examples

Example 1

With NDIVI2=5 in RUNSPEC keyword DIMENS:

<table>
<thead>
<tr>
<th>ADDZCORN</th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>10.0 3 4 3 4 1 5</td>
<td>/ Add 10ft to all layers</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>-20.0 7 9 6 9 1 5 7 9 6 9</td>
<td>/ Raise a fault block 20ft</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>-15.0 10 13 15 1 1 4* TOP</td>
<td>/ Raise the top surface only</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Example 2**

Add 10 ft. to the top-left corner of cells (3,3,1) to (3,3,5):

```
ADDZCORN
10.0  3  0  3  0   1 5     /
```

**Example 3**

Add 10 ft. to the lower-left corner of cell (3,3,1), while only modifying the top corner depth and leaving the adjacent cells unmodified.

```
10.0  3  0  0  3   1 1     3 3   3 3   'TOP' /
```
**ADSALNOD**  Salt concentration nodes for polymer solution adsorption

The data comprises NTSFUN (see RUNSPEC keyword `TABDIMS`) tables of salt concentration values, each terminated by a slash (/), describing the nodal values for salt concentration to be used in the calculation of the polymer solution adsorption.

Note that this differs from the `SALTNODE` keyword in being specified by saturation function region rather than by PVT region.

Each table consists of up to NSSFUN (see RUNSPEC keyword `TABDIMS`) values of the salt concentration. Within each table the values should be positive and strictly increasing.

**UNITS:** kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/cc (LAB)

The number of entries in each table must correspond to the number of polymer solution adsorption entries used in the `PLYADSS` keyword.

Note that if keyword `ECLMC` is selected, the `BRINE` keyword activates the multi-component brine model (ECLIPSE 100 only) and functionality such as the polymer salt-sensitivity option (keywords `POLYMER`, `BRINE`, `SALTNODE` and `ADSALNOD`) cannot be used.

**Example**

With NTSFUN=2 and NSSFUN \( \geq 4 \):

```
ADSALNOD
  1.0
  5.0
  10.0
  30.0 /
  5.0
  30.0 /
```
Generic adsorption functions

This keyword supplies coefficients for analytical adsorption functions which can be used together with the polymer, surfactant, alkaline, foam and environmental tracer options (see the RUNSPEC keywords POLYMER, SURFACT, ALKALINE, FOAM and TRACERS, respectively). The keyword can be used in place of POLYADS, SURFADS, ALKADS, FOAMADS and TRADS. The analytical adsorption function allows for dependencies of adsorption on rock permeability as well as brine salinity in the case of polymer and surfactant (see "Brine tracking" in the ECLIPSE Technical Description for information on the calculation of standard and effective salinities for polymer and surfactant).

Currently a generalized Langmuir adsorption function is available in the ADSORP keyword:

\[
C_{ads} = \frac{aC^m}{1 + bC} = (a_1 + a_2C_{SE}) \left( \frac{K_{ref}}{K} \right)^n,
\]

where

- \(C_{ads}\) denotes the rock adsorbed concentration
- \(C\) denotes the concentration in the solution surrounding the rock
- \(m\) denotes an exponent for concentration dependence
- \(C_{SE}\) denotes the (effective) salinity for polymer/surfactant
- \(K\) denotes the grid block permeability \(K = K_x K_y K_z\)
- \(K_{ref}\) denotes the reference permeability
- \(a_1\) denotes an adsorption coefficient
- \(a_2\) denotes an adsorption coefficient for salinity dependence
- \(b\) denotes an adsorption coefficient
- \(n\) denotes an exponent for permeability dependence

The data comprises NTSFUN+1 (see item 1 in the RUNSPEC keyword TABDIMS) records of data, each terminated by a slash (/).

First record

The first record holds the name of the adsorbing component. Valid options are: POLYMER, SURFACT, ALKALINE, FOAM or the three-character name of an environmental tracer.

Remaining records

The following NTSFUN records hold adsorption coefficients for each region:

1. Name of adsorption isotherm. This item must be set to LANGMUIR which is currently the only adsorption function available.

   DEFAULT: LANGMUIR.
2. Adsorption coefficient $a_1$.

DEFAULT: no default.

UNITS: $(\text{sm}^3/\text{kg})^m$ (METRIC), $(\text{stb}/\text{lb})^m$ (FIELD), $(\text{scc}/\text{gm})^m$ (LAB)

If ADSORP is used with an environmental tracer and a unit has been specified for this tracer in keyword TRACER, the units employed here are:

UNITS: $(\text{sm}^3)^m/(\text{TR}^{m-1} \text{ kg})$ (METRIC), $(\text{stb})^m/(\text{TR}^{m-1} \text{ lb})$ (FIELD), $(\text{scc})^m/(\text{TR}^{m-1} \text{ gm})$ (LAB)

where TR is the unit supplied for the tracer.

3. Adsorption coefficient for salinity dependence $a_2$. This item is only relevant when the ADSORP keyword is used in conjunction with the polymer or surfactant option. In all other cases the item is ignored.

For surfactant the effective salinity for surfactant is applied in the adsorption calculation (see keyword SURFESAL as well as the "Brine tracking" section in the ECLIPSE Technical Description). For polymer the salinity applied in the adsorption calculation depends on the type of brine model. When used with the Multi-Component Brine model the effective salinity for polymer (see keyword PLYESAL) will be used. Otherwise the standard brine concentration will be applied. Surfactant adsorption only allows a dependence on the effective salinity for surfactant, as activated by the SURFESAL keyword, and not the standard salinity.

DEFAULT: 0.0

When used with an effective salinity, as measured in moles equivalent per unit standard volume, the units of $a_2$ are

UNITS: $(\text{sm}^3)^{m+1}/(\text{kg}^m \text{ kg-M})$ (METRIC), $(\text{stb}^{m+1})/(\text{lb}^m \text{ lb-M})$ (FIELD), $(\text{scc})^{m+1}/(\text{gm}^m \text{ gm-M})$ (LAB)

When used with the standard brine concentration, as measured in mass per unit standard volume, the units of $a_2$ are

UNITS: $(\text{sm}^3/\text{kg})^{m+1}$ (METRIC), $(\text{stb}/\text{lb})^{m+1}$ (FIELD), $(\text{scc}/\text{gm})^{m+1}$ (LAB)

**Note:** Adsorption is always reversible with salinity, even when it is irreversible in concentration.

4. Adsorption coefficient $b$.

DEFAULT: no default

UNITS: $\text{sm}^3/\text{kg}$ (METRIC), $\text{stb}/\text{lb}$ (FIELD), $\text{scc}/\text{gm}$ (LAB)

If ADSORP is used with an environmental tracer and a unit has been specified for this tracer in keyword TRACER, the units employed here are:

UNITS: $\text{sm}^3/(\text{TR})$ (METRIC), $\text{stb}/(\text{TR})$ (FIELD), $\text{scc}/(\text{TR})$ (LAB)

where TR is the unit supplied for the tracer.

5. Exponent, $m$, for concentration dependence.

DEFAULT: 1

UNITS: dimensionless
6. Exponent, \( n \), for permeability dependence.
   
   DEFAULT: 0.5
   
   UNITS: dimensionless

7. Reference permeability \( k_{ref} \). This is the permeability at which the adsorption coefficients are specified. Permeability dependencies can be ignored by setting item 6 to 0.
   
   DEFAULT: no default
   
   UNITS: mD (METRIC), mD (FIELD), mD (LAB)

---

**Note:** The adsorption function relates the solution concentration of a component per unit volume of host phase to the rock adsorbed concentration per unit mass of rock. The rock mass density as well as the adsorption index must be supplied in the respective rock keyword, that is `PLYROCK` for polymer adsorption and `TRROCK` for tracer adsorption for example.

---

**Note:** The `ADSORP` keyword with `POLYMER` as an item and the `PLYADS` keyword are mutually exclusive and similarly for `ADSORP` with `SURFACT`, `ALKALINE`, `FOAM` or an environmental tracer as an item and the `SURFADS`, `ALKADS`, `FOAMADS` and `TRADS` keywords.

---

**Example**

Surfactant/polymer run with one environmental tracer ET1 and NTSFUN = 2:

```plaintext
ADSORP POLYMER /
   -- isotherm a1 a2 b m n k_ref
   LANGMUIR 0.0012 0.7 0.4 1.1 0.5 100/
   LANGMUIR 0.0009 0.5 0.1 1.2 0.3 50/
ADSORP SURFACT /
   -- isotherm a1 a2 b m n k_ref
   LANGMUIR 0.0012 0.7 0.4 1* 0.5 100/
   / -- default second table to first table
ADSORP ET1 /
   -- isotherm a1 a2 b m n k_ref
   LANGMUIR 0.001 1* 0.2 1* 0.5 250/
   LANGMUIR 0.005 1* 1.1 1* 0.0 50/
```

---

**Keywords**

303
Use the AIM solution option

This keyword selects the AIM (Adaptive IMplicit) solution option for ECLIPSE 300. This is the recommended option for most studies, and is the default. Alternative options are FULLIMP (fully implicit), IMPSAT and IMPES (IMplicit Pressures, Explicit Saturations). Generally AIM avoids the timestep restrictions imposed by small blocks, particularly those containing wells, whilst not suffering the much greater computational expense of a fully implicit solution.

For Thermal simulations the FULLIMP method is the default.

For BLACKOIL simulations the FULLIMP method is the default.

See "Formulation of the equations" in the ECLIPSE Technical Description.

Note: For ECLIPSE 300, the maximum rate of change of capillary pressure in explicit cells specified by the DPCDT keyword, which is assumed to be active by default for the AIM solution method, should be reviewed and checked for compatibility with the magnitudes of the oil-water and gas-oil capillary pressures specified via the saturation functions.
AIMCON

Controls for AIM

The data consists of some or all of the following data items, which control the implicit cell choice within an AIM run. The data must be terminated by a slash (/).

1. An integer which controls the implicit treatment of the wells.
   - 0 Wells are not given any special treatment.
   - 1 All the inner solver direction lines of cells containing the completions of open wells are set to be implicit.
   - 2 All the cells containing the completions of open wells are set to be implicit.
   - 3 All the cells containing the completions of open wells and their neighbors are set to be implicit.

   DEFAULT: 2
   Thermal DEFAULT: 3

2. An integer which controls the implicit treatment of LGRs (this item does not override the NOAIMLGR keyword).
   - 0 All cells in LGRs are set implicit
   - 1 LGRs are not given any special treatment.

   DEFAULT: 1

3. All cells below the specified pore volume threshold are made implicit. A negative value indicates that no pore volume limit is enforced.

   UNITS: \text{m}^3 (\text{METRIC}), \text{rb (FIELD)}, \text{rcc (LAB)}, \text{rm}^3 (\text{PVT-M})

   DEFAULT: -1.0

4. An integer specifying the number of timesteps taken before reevaluating the AIM state.

   The AIM state is always reevaluated at the beginning of a new report step.

   If this item is negative (and item 5 is greater than zero), the AIM state is always reevaluated if throughput or solution change is limiting the current step.

   DEFAULT: 5

5. The fractional throughput target threshold.

   The AIM state is always reevaluated at the beginning of a new report step.

   All cells (up to the maximum % specified in item 6) with a throughput above the specified fraction of the current throughput target ratio (see item 8 of TSCRIT) are set to implicit.

   A negative value indicates that this selection method is not used; instead the pre-2000A AIM selection criteria are activated.

   DEFAULT: 0.5

   \textbf{Thermal option} \hspace{1cm} DEFAULT: 0.1

   \textbf{CO2STORE option} \hspace{1cm} DEFAULT: 0.01
6. The maximum percentage of cells that can be set to implicit when the throughput change is greater than the target set in item 5. This value must be between 0.0 and 100.0, and can be altered using the AIMFRAC keyword.

If the criterion specified in item 5 produces more than the specified limit, then all these cells are sorted by throughput change, and the top percentage (specified by this item) are chosen.

   DEFAULT: 5.0

   Thermal option
   Thermal option: DEFAULT: 100.0

   CO2STORE option
   DEFAULT: 100.0

7. The fractional solution change target threshold.

   All cells (up to the maximum % specified in item 8) with a normalized solution change above the specified fraction of the current solution change target ratio (see item 10 of TSCRIT) are set to implicit. A negative value turns off this selection criterion.

   DEFAULT: 0.125

   Thermal option
   DEFAULT: -1.0

8. The maximum percentage of cells that can be set to implicit when the solution change is greater than the target set in item 7. This value must be between 0.0 and 100.0.

If the criterion specified in item 7 produces more than the specified limit, then all these cells are sorted by solution change, and the top percentage (specified by this item) are chosen.

   DEFAULT: 5.0

   Thermal option
   DEFAULT: 100.0

9. An integer specifying the number of AIM state evaluations that an implicit cell remains implicit after being set, regardless of the selection criteria.

This option is only activated when item 5 is greater than 0.

   DEFAULT: 5

   Thermal option
   DEFAULT: (Maximum number of non-linears allowed)/2, for instance NEWTMX /2 where NEWTMX is defined via item 1, record 3 of the TUNING keyword.

10. The extrapolation factor to be applied to the solution change before applying the selection criteria controlled by items 5 and 7.

   DEFAULT: 1.0

   Thermal only

11. The fractional temperature change target threshold.

   All cells (up to the maximum% specified in item 12) with a temperature change above the specified fraction of the current temperature change target (see item 17 of TSCRIT) are set to implicit. A negative value turns off this selection criterion.

   DEFAULT: -1.0

   Thermal only

12. The maximum percentage of cells that can be set to implicit when the temperature change is greater than the target set in item 11. This value must be between 0.0 and 100.0.

If the criterion specified in item 11 produces more than the specified limit, then all these cells are sorted by solution change, and the top percentage (specified by this item) are chosen.

   Thermal only
Thermal only

13. Threshold for gas saturation above which a cell will be designated as implicit. A negative value turns this option off.
   DEFAULT: 100.0

Thermal only

14. Threshold for temperature difference above which a cell will be designated as implicit. A negative value turns this option off.
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
   DEFAULT: 1°C

If item 5 is less than zero, then the selection criteria for Implicit cells with an AIM run are compatible with pre-2000A releases.

The basic premise with these selection criteria is to extrapolate a vector of timestep controlling values $s$ to the next time-level:

$$S^{(n+1)} = S^{(n)} + \delta \times \Delta t^{(n)} \max \left[ \frac{(S^{(n)} - S^{(n-1)})}{\Delta t^{(n-1)}}, 0 \right]$$  \hspace{1cm} \text{Eq. 3.1}

Three criteria can be used in selecting AIM cells, $S$ can be either the throughput (used with items 5 and 6), the solution change (used with items 7 and 8) or the temperature change (used with items 11 and 12). $\delta$ is the extrapolation factor (specified in item 10). A cell $i$ is set implicit if $S_i > \beta \times \text{target}$ where $\beta$ is the fraction specified in items 5, 7 or 11 and target is the timestepping target specified by items 8, 10 or 17 of the TSCRIT keyword. If the number of cells selected is above that specified by the limits imposed in items 6, 8 or 12, then the selected cells are sorted and only the largest values, up to the limit on the number of implicit cells, are chosen.

Note that if item 6 is set to 7% and item 8 is set to 3%, then up to 10% of the reservoir could be made implicit at any time (ignoring any well cells, that is item 1 is zero). As implicit cells will persist for a number of AIM state evaluations (item 9), then it is possible (although unlikely) that 100% of the reservoir could be made implicit after 10 AIM state evaluations.

**Example**

In this example up to 5% (item 6) of the reservoir could be made implicit if the time extrapolated throughput of a cell is greater than a half (item 5) of the target, and up to 5% (item 7) of the reservoir could be made implicit if the time extrapolated solution change is within an eighth (item 6) of the target. All cells made implicit remain implicit for 5 (item 9) AIM state evaluations, and the AIM states are evaluated after every timestep (item 3). The items specified in the example have been shown to give good performance over a wide range of simulations.

```
AIMCON
   --wells--lgr--minpV--#eval--TPT(thres)--TPT(max)--SCT(thres)--
   --SCT(max)--hist--exprap
   2  1*  1*  1  0.5  5  0.125  5  5  1
/  
```

**Keywords**
**Set target implicit fraction in AIM mode**

This keyword allows the user to modify the target fraction of cells which are implicit in an AIM (Adaptive Implicit Method) run. The default is 0.01, which is reasonable for most runs. Decreasing the target fraction will reduce the memory required and the CPU time per step, although it may reduce the size of each step. Increasing the target fraction has the opposite effect. Large `AIMFRAC` values should be avoided - `FULLIMP` may be preferable in this case.

This can also be altered using item 6 of keyword `AIMCON`. An additional default value for the Thermal option can be chosen.

DEFAULT: 0.01

**Note:** In the absence of the `AIMFRAC` keyword the default AIM fraction is 0.05 (5%) or as specified by the `AIMCON` keyword.

### Example

Set a target of 10 per cent of cells to be implicit:

```
AIMFRAC 0.1
```
**AIMPVI**

**Specify minimum pore volume for an explicit cell**

If the AIM solution method is being used, this keyword may be used to enforce a minimum reservoir pore volume below which a cell is always solved implicitly. This can be useful for radial studies, in which inner grid blocks, near the well, may be better locked in implicit mode.

The AIMPVI keyword may be specified more than once in the SCHEDULE section.

**UNITS:** \( \text{m}^3 \) (METRIC), \( \text{rb} \) (FIELD), \( \text{rcc} \) (LAB), \( \text{m}^3 \) (PVT-M)

**Example**

```plaintext
AIMPVI
12 /
```
**AJGPARAM Specify static parameters for adjoint gradient calculation**

*Note:* This keyword can only be used if the adjoint gradient option for sensitivities has been requested in the `RUNSPEC` section using the `AJGRADNT` keyword.

This keyword is used to specify a list of static grid properties for which the sensitivities of objective functions (specified using the `AJGWELLS` keyword) will be calculated at the end of the simulation run using the adjoint method (see “Adjoint gradient screening” in the *ECLIPSE Technical Description*). These sensitivities are calculated with respect to the specified static properties on every simulation grid cell at every report time, and they are output to the restart files. (Note that the sensitivities output to the restart file will be encoded unless the `UNCODHMD` keyword is present in the `RUNSPEC` section.)

This keyword should only appear once in the data file.

The keyword must be followed by a set of mnemonics, each specifying a parameter for which sensitivity calculations are required. The list should be terminated with a slash (/). An empty list is not permitted.

The available mnemonics are:

- `PORV` Pore volume
- `PORO` Porosity
- `TRANX` Transmissibility in the x-direction
- `TRANY` Transmissibility in the y-direction
- `TRANZ` Transmissibility in the z-direction
- `TRANXY` Transmissibility in the x- and y-directions (horizontal)
- `PERMX` Permeability in the x-direction
- `PERMY` Permeability in the y-direction
- `PERMZ` Permeability in the z-direction
- `PERMXY` Permeability in the x- and y-directions (horizontal)

**Example**

This example shows how to request sensitivities with respect to the pore volume and x-permeability in every simulation grid cell at each report time.

```
AJGPARAM 'PORV' 'PERMXY' /
```

---

*Keywords*

310
AJGRADNT Request sensitivities with respect to grid properties using the adjoint method

This keyword activates the option to perform calculations of sensitivities using the adjoint method. It requires no data.

These sensitivities are the response of objective functions (such as the flowing bottom-hole pressure in a particular well) to changes in static grid properties (such as pore volume in each simulation grid cell). The objective functions and static grid properties are defined in the SCHEDULE section using the AJGWELLS and AJGPARAM keywords respectively.

These sensitivities are output to the restart files; the values will be encoded unless the UNCODHMD keyword is present in the RUNSPEC section.

Note: This option is not currently available with the parallel option. See “Adjoint gradient screening” in the ECLIPSE Technical Description for a complete list of the requirements and limitations of this option.
**AJGWELLS**  Specify objective functions for adjoint gradient calculation

*Note:* This keyword can only be used if the adjoint gradient option for sensitivities has been requested in the **RUNSPEC** section using the **AJGRADNT** keyword.

This keyword is used to specify the objective functions for which sensitivities will be calculated at the end of the simulation run using the adjoint method (see "Adjoint gradient screening" in the **ECLIPSE Technical Description**). These sensitivities are calculated with respect to static properties (specified using the **AJGPARAM** keyword) on every simulation grid cell at every report time, and they are output to the restart files. (Note that the sensitivities output to the restart file will be encoded unless the **UNCODHMD** keyword is present in the **RUNSPEC** section.)

The keyword must be followed by a set of records, each specifying a parameter to be used in the optimization. Each record should be terminated with a slash (/) and the set of records must end with a slash. At least one record is required for the calculation of sensitivities; an empty record is not allowed. (The total number of records should not exceed 200.)

1. **Well name.**
   This should be the name of a well that has already been, or will be, defined in the **SCHEDULE** section. It is not possible to use wildcard characters to specify more than one well.

2. **Objective function mnemonic.**
   The list of available keywords is:

   - **WBHP** Flowing bottom hole pressure
   - **WOPR** Oil production rate
   - **WGPR** Gas production rate
   - **WGIR** Gas injection rate
   - **WWPR** Water production rate
   - **WWIR** Water injection rate
   - **WLPR** Liquid production rate

**Example**

This example shows how to request sensitivities for a selection of objective functions on a selection of wells.

```
AJGWELLS
 'P1'  'WWPR'  /
 'P1'  'WBHP'  /
 'P2'  'WLPR'  /
 'I1'  'WWIR'  /
 'I1'  'WBHP'  /
 /
```

**Keywords**

312
**Alkaline adsorption functions**

The data comprises NTSFUN (see item 1 of keyword TABDIMS) tables of alkaline adsorption functions, each terminated by a slash (/), describing the adsorption of alkaline by the rock formation. Each table consists of the following columns of data:

1. The local alkaline concentration in the solution surrounding the rock.
   - The values should increase monotonically down the column.
   - **UNITS**: kg/sm³ (METRIC), lb/stb (FIELD), gm/sc (LAB)

2. The corresponding saturated concentration of alkaline adsorbed by the rock formation.
   - These values should increase monotonically down the column.
   - **UNITS**: kg/kg (METRIC), lb/lb (FIELD), gm/gm (LAB)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see item 3 of keyword TABDIMS). If any table after the first is defaulted, then that table is copied from the previous table.

**CAUTION:** The first row in the adsorption tables should consist of a pair of zeroes.

See "Alkaline model" in the *ECLIPSE Technical Description* for further information.

**Example**

With NTSFUN=2 and NSSFUN≥5

```plaintext
ALKADS
 .0000 .00000
 .0003 .00005
 .0005 .00007
 .0008 .00008
 .0010 .00008 /
 .0000 .00000
 .0005 .00007
 .0008 .00008
 .0010 .00010 /
```
**ALKALINE**

**Enables the Alkaline model**

This indicates that the Alkaline Model is required (see "Alkaline model" in the *ECLIPSE Technical Description*).

The keyword has no associated data.

---

*Note:* In the current version of ECLIPSE this feature is targeted at ASP scenarios and would often be referred to as a high pH model.

---

---

<table>
<thead>
<tr>
<th>x</th>
<th>ECLIPSE 100</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>ECLIPSE 300</td>
</tr>
<tr>
<td></td>
<td>SPECIAL</td>
</tr>
<tr>
<td></td>
<td>RUNSPEC</td>
</tr>
<tr>
<td></td>
<td>GRID</td>
</tr>
<tr>
<td></td>
<td>EDIT</td>
</tr>
<tr>
<td></td>
<td>PROPS</td>
</tr>
<tr>
<td></td>
<td>REGIONS</td>
</tr>
<tr>
<td></td>
<td>SOLUTION</td>
</tr>
<tr>
<td></td>
<td>SUMMARY</td>
</tr>
<tr>
<td></td>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>

---

---
## ALKROCK

### Specifies the alkaline-rock properties

The data comprises NTSFUN (see item 1 of keyword TABDIMS) records specifying the rock adsorption index for alkaline adsorption. Each record should be terminated by a slash (/).

Possible values for the adsorption index are 1 or 2.

If a value of 1 is selected, then the alkaline adsorption isotherm is retraced whenever the local alkaline concentration in the solution decreases.

If a value of 2 is selected, then no alkaline desorption may occur and the alkaline effect on Polymer or Surfactant adsorption is assumed to be permanent: The multiplier tables in ALSURFAD or ALPOLADS are not retraced when the alkaline concentration decreases.

If any record after the first is defaulted, then that record is copied from the previous record.

**Note:** If the Polymer Flood Model is being used, the rock mass density required in the adsorption calculation is taken from the PLYROCK keyword. Otherwise, if the Surfactant Flood Model is used, the rock mass density is taken from the SURFROCK keyword.

See "Alkaline model" in the ECLIPSE Technical Description for further information.

### Example

With NTSFUN=2

```
ALKROCK
   2  /
   1  /
```

<table>
<thead>
<tr>
<th>Keywords</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>ECLIPSE 100</td>
<td>ECLIPSE 300</td>
<td>SPECIAL</td>
</tr>
<tr>
<td>RUNSPEC</td>
<td>GRID</td>
<td>EDIT</td>
</tr>
<tr>
<td>PROPS</td>
<td>REGIONS</td>
<td>SOLUTION</td>
</tr>
<tr>
<td>SUMMARY</td>
<td>SCHEDULE</td>
<td></td>
</tr>
</tbody>
</table>
**Transport coefficient tables**

The ALPHA keyword should provide NTALPHA tables of transport coefficients for oil and gas component flows as functions of one of the total mole fractions (see item 16 of keyword TABDIMS and also "Transport coefficients" in the ECLIPSE Technical Description). The component mole fraction to be used is identified separately with the TRCOEF keyword.

Each line of each table contains the mole fraction, then $N_c$ oil transport coefficients, followed by $N_c$ gas transport coefficients.

**Example**

Example with 5 components and NTALPHA=2:

```
ALPHA
-- Oil coefficients    Gas coefficients
  0  1.5  1.2  1  1  1     2.5  1  1  1  1
  1  1  1  1  1  1     1  1  1  1  1
  / 0  1.4  1.1  1  1  1     2.4  1  1  1  1
  / 1  1  1  1  1  1     1  1  1  1  1
```
**ALPHAD**

Drainage transport coefficient tables

The **ALPHAD** keyword is used to modify component mobilities for drainage (decreasing wetting phase saturation) processes only when hysteresis is occurring. Hysteresis should be specified using **SATOPTS** keyword with HYSTER option and also the keyword **EHYSTR**.

The **ALPHAD** tables are input for drainage processes as functions of one of the component mole fractions. This component mole fraction is identified separately with the **TRCOEF** keyword.

The **ALPHAD** keyword should provide NTALPHA tables of transport coefficients (see item 16 of keyword **TABDIMS** and also "Transport coefficients" in the **ECLIPSE Technical Description**). Each line of each table contains the mole fraction, then N\textsubscript{c} oil transport coefficients, followed by N\textsubscript{c} gas transport coefficients.

It is also possible to modify the component mobilities for imbibition processes with the keyword **ALPHAI**.

**Example**

Example with 5 components, NTALPHA=2 and component number 3 specified by **TRCOEF**:

```
ALPHAD
   --Z3 Oil coefficients   Gas coefficients
     0   1.5 1.2 1 1 1  2.5 1 1 1 1
     1   1   1    1 1 1  1   1 1 1 1
     / 0   1.4 1.1 1 1 1  2.4 1 1 1 1
     / 1   1   1    1 1 1  1   1 1 1 1
```

---

ECLIPSE Reference Manual

Keywords

317
**Imbibition transport coefficient tables**

The `ALPHAI` keyword is used to modify component mobilities for imbibition (increasing wetting phase saturation) processes only when hysteresis is occurring. Hysteresis should be specified using `SATOPTS` keyword with `HYSTER` option and also the keyword `EHYSTR`.

The `ALPHAI` tables are input for imbibition processes as functions of one of the component mole fractions. This component mole fraction is identified separately with the `TRCOEF` keyword.

The `ALPHAI` keyword should provide `NTALPHA` tables of transport coefficients (see item 16 of keyword `TABDIMS` and also "Transport coefficients" in the *ECLIPSE Technical Description*). Each line of each table contains the mole fraction, then `N_c` oil transport coefficients, followed by `N_c` gas transport coefficients.

It is also possible to modify the component mobilities for drainage processes with the keyword `ALPHAD`.

**Example**

Example with 5 components, `NTALPHA=2` and component number 3 specified by `TRCOEF`:

```
ALPHAI

Oil coefficients  Gas coefficients
0 1.5 1.2 1 1 1 2.5 1 1 1 1
1 1 1 1 1 1 1 1 1 1
/
0 1.4 1.1 1 1 1 2.4 1 1 1 1
1 1 1 1 1 1 1 1 1 1
/ 
```
ALPHANUD  ALPHAD region numbers

The keyword should be followed by one integer for every grid block in the current input box specifying the ALPHAD region to which it belongs. The region number should not be less than 1 or greater than the number of ALPHAD tables, NTALPHA (item 16 in keyword \texttt{TABDIMS}). The data must be terminated by a slash (/).

The \texttt{ALPHANUD} region number specifies which set of transport coefficient tables (input using \texttt{ALPHAD} in the \texttt{PROPS} section) should be used to calculate transport coefficients for drainage in a compositional model.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

**Example**

With \texttt{NTALPHA}=4; \texttt{NDIVIX}=8, \texttt{NDIVIY}=6 and \texttt{NDIVIZ}=5 specified in \texttt{RUNSPEC keyword DIMENS}; and no input \texttt{BOX} set:

\begin{verbatim}
ALPHANUD  
8*1 16*2 8*3 16*4  
8*1 16*2 8*3 16*4  
8*1 16*2 8*3 16*4  
8*1 16*2 8*3 16*4  
8*1 16*2 8*3 16*4  
8*1 16*2 8*3 16*4  
/ 
\end{verbatim}
### ALPHANUI

**ALPHAI region numbers**

The keyword should be followed by one integer for every grid block in the current input box specifying the ALPHAI region to which it belongs. The region number should not be less than 1 or greater than the number of ALPHAI tables, NTALPHA (item 16 in keyword TABDIMS). The data must be terminated by a slash (/).

The ALPHANUI region number specifies which set of transport coefficient tables (input using ALPHAI in the PROPS section) should be used to calculate transport coefficients for imbibition in a compositional model.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

#### Example

With NTALPHA=4; NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS; and no input BOX set:

```
ALPHANUI
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4 /
```

---

**Keywords**

- ECLIPSE
- SPECIAL
- PROPS
- REGIONS
- SOLVENT
- SUMMARY
- SCHEDULE

---

ECLIPSE Reference Manual

320
ALPHANUM  ALPHA region numbers

The keyword should be followed by one integer for every grid block in the current input box specifying the ALPHA region to which it belongs. The region number should not be less than 1 or greater than the number of ALPHA tables, NTALPHA (item 16 in keyword TABDIMS). The data must be terminated by a slash (/).

The ALPHA region number specifies which set of transport coefficient tables (input using ALPHA in the PROPS section) should be used to calculate transport coefficients in a compositional model.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

Example

With NTALPHA =4: NDIVIX =8, NDIVIY =6 and NDIVIZ =5 specified in RUNSPEC keyword DIMENS; and no input BOX set:

```
ALPHANUM
 8*1 16*2 8*3 16*4
 8*1 16*2 8*3 16*4
 8*1 16*2 8*3 16*4
 8*1 16*2 8*3 16*4
 8*1 16*2 8*3 16*4 /
```
Polymer adsorption multipliers as a function of alkaline concentration

The data comprises NTSFUN (see item 1 of keyword TABDIMS) tables of polymer adsorption multipliers as a function of the alkaline concentration, each terminated by a slash (/). Each table consists of 2 columns of data:

1. The local alkaline concentration.
   The values should increase monotonically down the column.
   UNITS: kg/sm³ (METRIC), lb/stb (FIELD), gm/sec (LAB)

2. The corresponding polymer adsorption multiplier.
   UNITS: dimensionless

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see item 3 of keyword TABDIMS). If any table after the first is defaulted, then that table is copied from the previous table.

CAUTION: The first row in each table should consist of a multiplier of one associated to a concentration of zero.

See "Alkaline Model" in the ECLIPSE Technical Description for further information.

Example

With NTSFUN = 2 and NSSFUN ≥ 5

```
ALPOLADS
.0000  1.0000
.0003  0.90005
.0005  0.90007
.0008  0.90008 /
.0010  0.90008 /
.0000  1.00000
.0010  0.90010 /
```
ALSURFAD Surfactant adsorption multipliers as a function of alkaline concentration

The data comprises NTSFUN (see item 1 of keyword TABDIMS) tables of surfactant adsorption multipliers as a function of the alkaline concentration, each terminated by a slash (/). Each table consists of 2 columns of data:

1. The local alkaline concentration.
   The values should increase monotonically down the column.
   UNITS: kg/sm³ (METRIC), lb/stb (FIELD), gm/scce (LAB)

2. The corresponding surfactant adsorption multiplier.
   UNITS: dimensionless

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see item 3 of keyword TABDIMS). If any table after the first is defaulted, then that table is copied from the previous table.

**CAUTION:** The first row in each table should consist of a multiplier of one associated to a concentration of zero.

See "Alkaline model" in the ECLIPSE Technical Description for further information.

**Example**

With NTSFUN = 2 and NSSFUN ≥ 5

```
ALSURFADS
.0000  1.0000
.0003  0.90005
.0005  0.90007
.0008  0.90008 /
.0010  0.90008 /
.0000  1.00000
.0010  0.90010 /
```

**Keywords**

ALSURFADS
ALSURFST  Water/oil surface tension multipliers as a function of alkaline concentration

The data comprises NTPVT (see item 2 of keyword TABDIMS) tables of surface tension multipliers as a function of the alkaline concentration, each terminated by a slash (/). Each table consists of 2 columns of data:

1. The local alkaline concentration.
   The values should increase monotonically down the column.
   UNITS: kg/sm³ (METRIC), lb/stb (FIELD), gm/sc (LAB)

2. The corresponding surface tension multiplier.
   UNITS: dimensionless

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see item 4 of keyword TABDIMS). If any table after the first is defaulted, then that table is copied from the previous table.

The multiplier has no effect once the minimal surface tension in the SURFST table has been reached.

**CAUTION:** The first row in each table should consist of a multiplier of one associated to a concentration of zero.

See "Alkaline model" in the ECLIPSE Technical Description for further information.

**Example**

With NTPVT =2 and NPPVT ≥ 5

```plaintext
ALSURFST
.0000  1.0000
.0003  0.90005
.0005  0.90007
.0008  0.90008
.0010  0.90008 /
.0000  1.00000
.0010  0.90010 /
```

Keywords
**AMALGAM Specify LGR amalgamations**

The `AMALGAM` keyword is used to define groups of LGRs which are to be amalgamated.

This keyword can be followed by any number of records. Each record is of arbitrary length and is terminated by a slash (/). The set of records must be terminated with a blank record containing just a slash.

Each record defines one amalgamation, and should consist of the names of all the LGRs in the amalgamation (up to 8 characters). An LGR name root, ending with an asterisk (*) and enclosed in quotes, can be used to refer to several LGRs at once.

LGRs within the same amalgamation are allowed to touch, but must not overlap and may not be nested within each other. ECLIPSE treats an amalgamation as a single entity, with non-neighbor connections between any LGRs which touch.

**Note:** Additional non-neighbor connections may arise owing to domain decomposition in parallel runs and may require the inclusion of additional LGRs in an amalgamation.

**Note:** Non-neighbor connections must not be disabled by the keyword `NONNC` in the RUNSPEC section.

This facility allows you to refine the grid more efficiently around deviated wells. Wells may be completed in several local grids, as long as they belong to the same amalgamation.

For these wells, it is necessary to use the keywords `COMPLMPL`, `WELOPENL` and `WFRICTNL` in place of `COMPLUMP`, `WELOPEN` and `WFRICTN`. (`COMPLMPL` lumps connections into completions; `WELOPENL` shuts or reopens well connections; and `WFRICTNL` permits the modeling of friction wells completed across several local grids if the Wellbore Friction Option is active.)

If LGRs that do not touch are amalgamated, ECLIPSE still lumps them together as a single entity in the solution process. Thus, their local timesteps are synchronized.

**Note:** Only Cartesian LGRs may be amalgamated.

VE and dispersed flow models must be used in all or none of the LGRs in an amalgamation. The options under keyword `QMOBIL` must be the same for all LGRs within an amalgamation.

The maximum number of wells allowed in any amalgamation is the sum of the maxima specified in item 11 of keyword `CARFIN` for all the local grids in the amalgamation.

Since an amalgamation of local grids is solved as a single entity in ECLIPSE, `LGRON` and `LGROFF` actions for any local grid in an amalgamation apply to the whole amalgamation.

This keyword is not required by ECLIPSE 300. Connections are generated between nested and adjoining local grids in ECLIPSE 300 in all runs.

See "Amalgamating Cartesian local grids (ECLIPSE 100)" in the ECLIPSE Technical Description.

See also keywords `COMPDATL`, `WELOPENL`, `COMPLMPL`, `WPIMULTL` and `WFRICTNL`.

**Note:** When using the Flux Boundary Conditions option (see keyword `USEFLUX`) all local grid refinements within an amalgamation must be in the same flux region (see keyword `FLUXNUM`).
Note: When using the flux boundary conditions option (see keyword USEFLUX) in parallel, amalgamations may not be used outside of the USEFLUX region.

Example

Two local grid amalgamations:

```
AMALGAM 'NORTH*' /
  LGR1 LGR2 LGR3 LGR4 /
/
```
**AMF**

**Specifies cell initial aqueous composition explicitly**

This keyword can be used to specify the initial aqueous phase composition values for each cell in runs with the GASWAT, CO2SOL, H2SSOL, GASSOL or the thermal KWTABTn option active.

In the GASWAT case, the syntax follows that of XMF and YMF with the data entered as $N_x \cdot N_y \cdot N_z \cdot N_c$ values in normal order.

In the THERMAL case the input follows the syntax of XMF and YMF with data entered as $N_x \cdot N_y \cdot N_z \cdot (N_c + 1)$. Please note that when using the KWTABTn option or using the CO2STORE option together with THERMAL it is possible to specify H2O in the list of components set by COMPS, using the name H2O, in this case $N_x \cdot N_y \cdot N_z \cdot N_c$ values are entered.

In the CO2SOL case only two components, CO$_2$ and water, are present in the aqueous phase; hence only $N_x \cdot N_y \cdot N_z \cdot 2$ values are required with the CO$_2$ data followed by the water fraction. Usually it is preferable to use the RSW keyword to input the CO$_2$ concentration in the aqueous phase.

In the H2SSOL case only two components, H$_2$S and water, are present in the aqueous phase; hence only $N_x \cdot N_y \cdot N_z \cdot 2$ values are required with the H$_2$S data followed by the water fraction.

If both CO2SOL and H2SSOL are present three components, CO$_2$, H$_2$S and water, are present in the aqueous phase; hence only $N_x \cdot N_y \cdot N_z \cdot 3$ values are required with the CO$_2$ data followed by H$_2$S data followed by the water fraction.

If GASSOL is present, NCMPHW components and water are present in the aqueous phase; hence only $N_x \cdot N_y \cdot N_z \cdot NCMPHW$ values are required with the NCMPHW data followed by the water fraction.
**AMFVD**

**Aqueous mole fractions versus depth**

This keyword can be used in a THERMAL run together with the CO2STORE option and/or when water $K_w$ values are defined using KWTABTn.

The data comprises a table of aqueous mole fractions (for example, CO$_2$, H$_2$S and H$_2$O) versus depth. Each table consists of the number of components plus 1 columns of data and must be terminated by a slash (/). Note that in a thermal run the H$_2$O component also should be specified.

When water-oil contact and gas-oil contact is above reservoir top then COMPVD and ZMFVD are not needed, AMFVD would be enough for initialization.

Column 1  
Depth values.  
Values should increase monotonically.  
UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

Columns 2 to $N_c + 1$  
Corresponding mole fraction for each component including water (H$_2$O).

If AMFVD is not used in equilibrated cases (EQUIL keyword present) then water will be entirely composed of H$_2$O.

The initial aqueous mole fractions will be set from this keyword, unless the concentration exceeds the saturated value in which case the saturated value will be taken. Note, however, that the aqueous concentration may not be in equilibrium with the gaseous concentration. The concentration reported at initialization is the concentration after flashing the system, and hence dissolved gases (for example, CO$_2$ or H$_2$S) may be redistributed between the phases in each grid block.

**Example**

Aqua mole fractions versus depth, one component and water.

```
AMFVD
----- gas water
100  0.001 0.999
10000 0.002 0.998/
```
API tracking option is enabled

This indicates that the API Tracking option is required (see "API tracking" in the ECLIPSE Technical Description).

This option can be used in both live and dead oil systems. The keyword RSCONSTT cannot be used in API Tracking problems.

The keyword has no associated data.
APIGROUP  Allows grouping of oil PVT tables for API tracking

The keyword should be followed by a single integer, terminated by a slash. The integer represents the maximum number of groups of oil PVT tables available when the API Tracking option is active. The default is 1.

The PVT tables are input in the usual way. Either the surface densities are set with keyword DENSITY, or the gravities are set with keyword GRAVITY. Normally the oil surface densities should increase monotonically with the table number (or the oil API gravities should decrease monotonically) when the API tracking option is in use. However, if keyword APIGROUP is present, a break in the required monotonicity is taken to indicate that the subsequent tables belong to the next API group. Thus the set of oil PVT tables can be arranged into API groups by breaking the monotonicity of the oil surface density or gravity. Within each API group, the densities increase (or gravities decrease) monotonically. An error is flagged if the total number of API groups exceeds the maximum number specified in this keyword.

The separate groups of oil PVT tables can then be used in different areas of the reservoir. The REGIONS section keyword PVTPNUM is used to specify which API group is to be used for each grid block. For example, a cell in PVTPNUM region 2 will use the second API group to obtain the oil properties. For water and gas PVT properties, the PVTPNUM regions correspond to the actual table numbers, as in runs that do not use API tracking. So a cell in PVTPNUM region 2 still uses table number 2 for its water and gas properties.

Example

```plaintext
APIGROUP
  2 /
```

Keywords

330
API limiting

This keyword can be used to monitor the numbers of grid blocks with API values outside the expected range when the API Tracking option has been selected (see “API tracking” in the ECLIPSE Technical Description). In addition, the API values may optionally be constrained to lie within a specified range. This keyword consists of the following items:

1. Type of limiting to be applied.
   - NONE  Deselects any previously specified API value monitoring or limiting.
   - WARN  Provides a summary of the numbers of grid blocks with API values outside the specified range at each timestep and a more detailed tabulated summary at each report step.
   - LIMIT  Constrains the API values of grid blocks outside the specified range to the lower or upper limit as appropriate.
   - BOTH  Provides a summary of the numbers of grid blocks with API values outside the specified range before limiting at each timestep and a more detailed tabulated summary at each report step. It also constrains the API values to the lower or upper limit as appropriate at each timestep.

   DEFAULT: NONE

   Items 2 to 5 inclusive, which are defined below, are ignored and need not be specified if the first data item is set to NONE.

2. Scope of limiting to be applied.
   - LOWER  Applies monitoring and/or limiting to the lower API limit only.
   - UPPER  Applies monitoring and/or limiting to the upper API limit only.
   - BOTH  Applies monitoring and/or limiting to both the lower and upper API limits.

   DEFAULT: BOTH

3. Lower API limit.
   - UNITS: °API
   - DEFAULT: The minimum API value inferred from initialization via equilibration or enumeration.

4. Upper API limit.
   - UNITS: °API
   - DEFAULT: The maximum API value inferred from initialization via equilibration or enumeration.

5. Number of rows of tabular output in the more detailed tabulated summary at each report step.
   - DEFAULT: 10

This facility may be used both to monitor the values of grid blocks which are outside the expected API limits and to constrain the API values to these limits. If monitoring is specified, brief summaries of the numbers of grid blocks whose values are below the lower limit and above the upper limit are provided at each timestep together with the associated lowest and highest API values respectively. If API limiting is specified, these summaries pertain to the values before limiting is applied. More information is provided at
each report step where tabulated lists are provided for grid blocks with API values which are below the lower limit or above the upper limit. The lower and upper values are tabulated in increasing and decreasing order respectively with the number of rows in each table specified by item 5.

**Note:** The detailed tabulated summary at each report step is only available for parallel runs if item 8 of the `PARAOPTS` keyword is set to a value greater than zero.

**Note:** Care should be exercised in selecting the API limits when applying limiting. In general, these should not be within the expected range of API values which might reasonably be expected to occur during the simulation.

### Example

```
APILIM BOTH BOTH 20.0 35.0 25 /
... TSTEP 365 /
... APILIM NONE /
```
APIVD

Oil API versus depth for API tracking equilibration

The data comprises NTEQUL (see the keyword EQLDIMS) tables of oil API gravity versus depth, one for each equilibration region. Each table must be terminated by a slash (/), and consists of 2 columns of data:

1. Depth values.
   The values should increase monotonically down the column.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT–)

2. The corresponding values of oil API gravity.
   UNITS: °API

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NDRXVD (see the keyword EQLDIMS).

Example

With NTEQUL =1 and NDRXVD ≥ 2:

```
APIVD
7000  40.1
9000  36.5 /
```
AQANCONL  Specifies local grid refinement connection data for analytic aquifers

This keyword connects analytic aquifers, declared using AQUCT, AQUFETP, AQUFLUX, AQUCHWAT and AQUCHGAS, to one or more reservoir cells located within local grid refinements.

The connection to the reservoir is set up by an arbitrary box defined by lower and upper I, J and K indices within the local grid refinement (refer to items 3-8 below). For each active grid block within the box, an aquifer connection is made to the face specified in item 9 if either of the following conditions is met:

• the face is on the outside of the reservoir, or
• the face adjoins an inactive cell.

Any number of records may be entered, each terminated with a slash (/). The set of records must be terminated with a blank record containing just a slash.

The items in each record are:

1. Aquifer identification number
   This should be a number between 1 and the maximum number of aquifers, set in the keyword AQUDIMS.

2. Local grid refinement name (up to 8 characters)

3. Lower I-location of connecting grid blocks

4. Upper I-location of connecting grid blocks

5. Lower J-location of connecting grid blocks

6. Upper J-location of connecting grid blocks

7. Lower K-location of connecting grid blocks

8. Upper K-location of connecting grid blocks

9. Index defining the face of the reservoir to which the aquifer connects (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS):
   I-   I = 1 face
   I+   I = NDIVIX of LGR face
   J-   J = 1 face
   J+   J = NDIVIY of LGR face
   K-   K = 1 face (top)
   K+   K = NDIVIZ of LGR face (bottom)

10. The aquifer influx coefficient.
   This determines the fraction of the total communication between the reservoir and the aquifer for each cell defined in the box above.

   The default for each cell in the box is its face area. If a non-default value is entered, the value is assigned to each cell within the box.
If a reservoir cell is defined more than once, its previous value for the aquifer influx coefficient is added to the present value. The exception is the case where you enter a zero value, which then sets the influx coefficient for all the cells in the box to zero.

The aquifer influx coefficient is not used if the aquifer is a constant flux aquifer (keyword AQUFLUX). A negative value is not accepted.

UNITs: m² (METRIC), ft² (FIELD), cm² (LAB), m² (PVT-M)

11. Aquifer influx coefficient multiplier.

The multiplier is applied to the influx coefficients of the aquifer - cell connections within the box.

For constant flux aquifers (keyword AQUFLUX) the multiplier is applied to the cell areas.

DEFAULT: 1.0

12. Allow aquifer connections to cell faces adjoining active cells?

either: YES or NO

If this data item is set to YES, the restriction that a connected face must not adjoin an active cell is lifted, allowing aquifer connections into the middle of the reservoir. This option should be used with care and is not recommended for general use.

DEFAULT: NO

ECLIPSE 100

The total number of grid blocks connected to any single aquifer must not exceed the value set by item 6 in the AQUUDIMS keyword in the RUNSPEC section. In addition, this keyword cannot be used with the keyword AQUFET.

Note: Each aquifer can only have one connection in any given grid block. When more than one connection in a grid block is specified for a single aquifer only the first will be connected.

ECLIPSE 100

The aquifer connections can be listed by setting item 12 in the RPTSOL keyword to 2 (mnemonic AQUFET, AQUCT or AQUANCON).

For both simulators, an array with the name AQUIFERA is written to the INIT file to illustrate which cells are connected to each aquifer. If a cell is connected to aquifer N then the number 2^(N-1) appears in the AQUIFERA array. If a cell is connected to multiple aquifers then the appropriate output numbers are summed. Similarly, for gas aquifers, an array AQUIFERG is written to the INIT file.

ECLIPSE 300

The aquifer connections can be listed by specifying the argument AQANCONL in the RPTSOL keyword.

Note: Each unique aquifer can only be connected to one grid. That is, an aquifer can only be connected to either the global grid or to a single local grid refinement.

Note: The keyword AQANCONL must not be used in restarted runs, as the aquifer specification data cannot be changed.

ECLIPSE 100

Note: This keyword cannot be used if the PARALLEL keyword is present.
Example

One Carter-Tracy aquifer, connected to one side of a 10 x 10 x 2 local grid refinement called LGR with one face where the aquifer influx coefficient is set to 1000.

```
AQUCT
  1 7000.0 4000.0 200.0 .3 1.0E-5 500.0 50.0 10.0 1 1 /
/
AQANCONL
  1 LGR 1 1 10 1 2 'I+' 1000 /
```

Keywords
AQANNC  Set connection values explicitly for analytic aquifers

This keyword enables a specified connection to be set up between grid cells and an analytic aquifer which is defined using the AQUCT, AQUFETP or AQUFLUX keyword. Normally ECLIPSE will calculate the aquifer connections from the data given in the AQUANCON or AQUFET keywords, but the AQANNC keyword allows for more generality in defining the connections.

The keyword consists of a line of data for each required analytic aquifer connection, specifying the cells to be linked and the required area for communication. The cell coordinates are specified in natural order.

Each line of data must be terminated by a slash (/). After the last connection is defined, an additional slash on a new line terminates the list.

The values in each data record are:

1. Aquifer number
2. I-index of first cell joined to the non-neighbor connection (IX)
3. J-index of first cell joined to the non-neighbor connection (IY)
4. K-index of first cell joined to the non-neighbor connection (IZ)
5. Connecting area value for the grid cell (AREA)

**UNITS:** m² (METRIC), ft² (FIELD), cm² (LAB), m² (PVT-M)

**DEFAULT:** 0.0

Each line of data must be terminated by a slash (/). After the last connection is defined, an additional slash on a new line terminates the list. It is possible to specify this keyword more than once.

**Example**

```
AQANNC
  IA IX IY IZ AREA
  1  2  3  5  2.0 /
  1  2  3  6  4.5 /
/
```

---

**Keywords**
Sets initial tracer concentrations for analytic aquifers

This keyword is used to specify the value of the initial tracer concentration in each of the analytic aquifers in the run.

To specify the value of the initial concentration of either a water phase tracer or hydrocarbon component tracer in each of the analytic aquifers, refer to the keywords `AQUCT`, `AQUFETP`, `AQUFLUX`, `AQUCHWAT`, and `AQUFET` (ECLIPSE 100 only).

For an `AQUCHGAS` aquifer, only the initial concentration value of a hydrocarbon component tracer should be specified.

The `AQANTRC` keyword can only appear after any of the keywords `AQUFET`, `AQUFETP`, `AQUCT`, `AQUFLUX`, `AQUCHWAT` or `AQUCHGAS` have been used to define the aquifer types.

If a tracer is not specified in the list of tracer names, a default concentration value of 0.0 is assumed.

The keyword is followed by any number of records, each containing the following items of data:

1. Aquifer identification number.
   This should be a number between 1 and the maximum number of aquifers, entered using the `AQUDIMS` keyword.

2. Name of the tracer.
   The tracer must have been previously defined to exist in the water phase (see keyword `TRACER` in the `PROPS` section).

3. Value of the initial tracer concentration in the aquifer.
   This item should not be less than 0.

Each data record ends with a slash (/), and the set of records with a blank record, containing just a slash (/).

**Note:** Referring to items 1 and 2 described above, if the keyword `AQUFET` has been used to specify Fetkovich aquifer data, the aquifer identification number refers to the order in which the aquifers are defined.

**Note:** If the aquifer is specified with keyword `AQUFET`, `AQUFETP`, `AQUCT`, `AQUFLUX`, `AQUCHWAT` or `AQUCHGAS`, the aquifer identification number refers to the identification number entered in the 1st item of the keyword. In addition, the name of the tracer should be specified in 1 to 3 characters.

See also "Tracer tracking" in the *ECLIPSE Technical Description* for further information.

**Examples**

**Example 1**

With 2 aquifers specified in the keyword `AQUDIMS`:

```
AQANTRC
  1 WT1 1.0 /
  1 WT2 0.0 /
```
Example 2

Set 60% concentration of TR6 in the first aquifer.

```
AQANTRC
1 TR6 0.6 /
```

```
A-coefficient terms for water component

In GASWAT mode the normal temperature dependence of the Peng-Robinson equation for the water component is modified to take the form:

\[ a^{1/2} = A_1 + A_2 T + A_3 T^{-3} \]  

Eq. 3.2

The defaults for \( A_1, A_2 \) and \( A_3 \) are described in detail in "GASWAT option" in the ECLIPSE Technical Description. The AQCOEF keyword allows these to be modified.

AQCOEF contains a data item for each value. If any are defaulted they take the precalculated values.

Example

| AQCOEF | 0.4530 | 1.0 | 3.4E-02 |

Keywords
**AQSTREAM**

**Set composition of the injected gas stream or water analytic aquifer**

For compositional runs, this keyword should be used to specify the gas composition of the injected gas stream in an `AQUCHGAS` aquifer. This keyword must be placed after the `AQUCHGAS` aquifer definition for a gas aquifer.

For the `GASWAT` and `CO2STORE` options, this keyword can be used to specify the water composition of an `AQUCHWAT`, `AQUCT`, `AQUFETP` or `AQUFLUX` water analytic aquifer. This keyword must be placed after the `AQUCHWAT`, `AQUCT`, `AQUFETP` or `AQUFLUX` definition for a water aquifer.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values. Defaults of value 0 can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Aquifer identification number. This should be a number between 1 and the maximum number of aquifers, entered using the keyword `AQUDIMS`.

2. \( a_1 \), the mole fraction of the first component.

3. \( a_2 \), the mole fraction of the second component.

... 

\( N_c + 1 \). \( a_{N_c} \), the mole fraction of the \( N_c \)th component.

\( N_c \) mole fractions should be specified, where \( N_c \) is the number of components in the run. The values are checked using the condition that the sum of the mole fractions must be 1.0

Once the gas stream is defined, it should be used together with the `AQUCHGAS` keyword.

**Note:** For Black Oil runs, there is no need to use the `AQSTREAM` keyword. It will be assumed that there are two components, oil and gas. ECLIPSE 300 will set up the necessary default values.

**Example**

```
AQSTREAM
  1 0.5 0.3 0.1 0.05 0.05 /
/
```

---

**Keywords**
AQSTREAW  Set composition of multi-component water analytic aquifer

For the multi-component water option (COMPW), this keyword must be used to specify the water composition of an AQUCHWAT, AQUCT, AQUFETP or AQUFLUX aquifer. This keyword must be placed after the AQUCHWAT, AQUCT, AQUFETP or AQUFLUX definition for a water aquifer.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values. Defaults of value 0 can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Aquifer identification number. This should be a number between 1 and the maximum number of aquifers, entered using the keyword AQUDIMS.

2. \( a_1 \), the mole fraction of the first component.

3. \( a_2 \), the mole fraction of the second component.

... ...

\( a_{N_w} \), the mole fraction of the \( N_w \)th component.

\( N_w \) mole fractions should be specified, where \( N_w \) is the number of water components in the run. The values are checked using the condition that the sum of the mole fractions must be 1.0

See "Water components" in the ECLIPSE Technical Description for further details of the water component option.

Notes

- The keyword AQSTREAM must be used instead of AQSTREAW in a GASWAT or CO2STORE run.
- If the keyword AQSTREAM is not used in a multi-component water run, then an error will be emitted to the PRINT file and by default the water composition of an analytic water aquifer will be made up entirely of the last water component. However if the last water component is a surfactant, polymer or alkaline component (see CWTYPE), then the default aquifer water composition will be the preceding water component.

Example

```
AQSTREAM
  1 0.5 0.3 0.1 0.05 0.05 /
/ 
```

Keywords
AQUALIST  Sets up lists of aquifers for output

The keyword AQUALIST may be used to construct lists of analytic aquifers, which are lists of aquifer numbers. A name should be given for the aquifer list. Output can be obtained for the sum of all the aquifers in the list, that is the aquifer lists act similarly to groups for the summary keywords associated with aquifers. Also AQUIFERL is output to the INIT file if one of the aquifer keywords has been used in RPTSOL. If a cell is connected to aquifer list \(N\) then the number \(2^{(N-1)}\) appears in the AQUIFERL array. If a cell is connected to multiple aquifers contained in aquifer lists, the appropriate output numbers are summed.

The maximum number of aquifer lists and the maximum number of aquifers to be associated with any one list are defined with items 7 and 8 of AQUDIMS.

Each list should have a unique name, consisting of up to eight characters. If a list name is repeated those aquifers are appended to the original list

1. The aquifer list name.
   This should consist of up to 8 characters, enclosed in quotes.
2. MXAAQL+1
   A list of aquifer identification numbers up to a maximum of MXAAQL (item 8 of AQUDIMS).

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Example

```
AQUALIST
  'NORTHEAS' 1 2 3/
  'SOTHWEST' 4 5 /
/ 
```
**AQUANCON**

**Specifies connection data for analytic aquifers**

AQUANCON connects analytic aquifers, declared using **AUCT, AQUFETP, AQUFLUX, AUCHWAT or AUCHGAS**, and **AQUFET** in ECLIPSE 100, to one or more reservoir cells.

The connection to the reservoir is set up by an arbitrary box defined by lower and upper I, J and K indices (refer to items 2-7 below). For each active grid block within the box, an aquifer connection is made to the face specified in item 8 if either of the following conditions is met:

- the face is on the outside of the reservoir, or
- the face adjoins an inactive cell.

**Note:** Care must be taken while considering the IJK box extents. Multiple thin layers of cells on the reservoir boundary are strongly recommended over bulky boxes covering inner regions. ECLIPSE does not check for aquifer connections inside the reservoir. As a result, any cell within the prescribed box may be attached an aquifer as long as its adjacent cell is inactive. There are various situations when cells may be inactive inside the reservoir, for example due to their small pore volume (see **MINPV**) or when a sector model (see **USEFLUX**) is surrounded by inactive cells of the full-field model.

Any number of records may be entered, each terminated with a slash (/). The set of records must be terminated with a blank record containing just a slash.

**ECLIPSE 100**

The keyword can be specified in the **SOLUTION** section.

**ECLIPSE 300**

The keyword can be specified in the **GRID** and **SOLUTION** sections.

The items in each record are:

1. Aquifer identification number
   - This should be a number between 1 and the maximum number of aquifers, set in the keyword **AQUDIMS**.
2. Lower I-location of connecting grid blocks
3. Upper I-location of connecting grid blocks
4. Lower J-location of connecting grid blocks
5. Upper J-location of connecting grid blocks
6. Lower K-location of connecting grid blocks
7. Upper K-location of connecting grid blocks
8. Index defining the face of the reservoir to which the aquifer connects (where **NDIVIX, NDIVIY** and **NDIVIZ** are specified in **RUNSPEC** keyword **DIMENS**):
   - I- or X- I or X = 1 face
   - I+ or X+ I or X = NDIVIX face
   - J- or Y- J or Y = 1 face
   - J+ or Y+ J or Y = NDIVIY face
   - K- or Z- K or Z = 1 face (top)
K+ or Z+  K or Z = NDIVIZ face (bottom)

9. The aquifer influx coefficient.

This determines the fraction of the total communication between the reservoir and the aquifer for each cell defined in the box above.

The default for each cell in the box is its face area. If a non-default value is entered, the value is assigned to each cell within the box.

If a reservoir cell is defined more than once, its previous value for the aquifer influx coefficient is added to the present value. The exception is the case where you enter a zero value, which then sets the influx coefficient for all the cells in the box to zero.

The aquifer influx coefficient is not used if the aquifer is a constant flux aquifer (keyword AQUFLUX).

A negative value is not accepted.

UNITS: m² (METRIC), ft² (FIELD), cm² (LAB), m² (PVT-M)

10. Aquifer influx coefficient multiplier.

The multiplier is applied to the influx coefficients of the aquifer - cell connections within the box.

For constant flux aquifers (keyword AQUFLUX) the multiplier is applied to the cell areas.

DEFAULT: 1.0

11. Allow aquifer connections to cell faces adjoining active cells?

   either: YES or NO

   If this data item is set to YES, the restriction that a connected face must not adjoin an active cell is lifted, allowing aquifer connections into the middle of the reservoir. This option should be used with care and is not recommended for general use.

   DEFAULT: NO

**ECLIPSE 100**

The total number of grid blocks connected to any single aquifer must not exceed the value set by item 6 in the AQUIDIMS keyword in the RUNSPEC section. In addition, this keyword cannot be used with the keyword AQUFET.

**Note:** Each aquifer can only have one connection in any given grid block. When more than one connection in a grid block is specified for a single aquifer only the first will be connected.

**ECLIPSE 300**

The aquifer connections can be listed by specifying the mnemonic AQUANCON in the RPTSOL keyword.

For both simulators, an array with the name AQUIFERA is written to the INIT file to illustrate which cells are connected to each aquifer. If a cell is connected to aquifer N then the number \(2^{(N-1)}\) appears in the AQUIFERA array. If a cell is connected to multiple aquifers then the appropriate output numbers are summed. Similarly, for gas aquifers, an array AQUIFERG is written to the INIT file.
Note: Each unique aquifer can only be connected to one grid. That is, an aquifer can only be connected to either the global grid or to a single local grid refinement.

Note: The keyword AQUANCON must not be used in restarted runs, as the aquifer specification data cannot be changed.

See also "Aquifer modeling facilities" in the ECLIPSE Technical Description.

Example

One Carter-Tracy aquifer, connected to two vertical sides of a 10 x 10 x 2 grid with one face where the aquifer influx coefficient is set to 1000.

```
AQUCT
  1 7000.0 4000.0 200.0 .3 1.0E-5 500.0 50.0 10.0 1 1 /
/
AQUANCON
  1 1 10 10 10 1 2 'J+' /
  1 10 10 1 10 1 2 'I+' 1000. /
/
```
AQUCHGAS  Specifies the property data for constant head/pressure gas aquifers

The number of analytic aquifers is specified using the AQUUDIMS keyword in the RUNSPEC section.

Each record specifies the properties of a separate constant head/pressure gas ‘aquifer’ (ranging from 1 to the number of analytic aquifers), and contains the following items of data:

1. Aquifer identification number
   This should be a number between 1 and the maximum number of aquifers, entered using the keyword AQUUDIMS.

2. Datum depth
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

3. Gas pressure in the aquifer at the datum depth
   UNITS: barsa (METRIC), psia (FIELD), atm (LAB), atma (PVT-M)

4. Aquifer productivity index
   (Total influx rate per unit pressure difference.)
   UNITS: sm$^3$/day/bars (METRIC), Mscf/day/psi (FIELD), scc/hr/atm (LAB), sm$^3$/day/atm (PVT-M)

5. Table number for gas pressure properties (for example see keyword PVTG).
   DEFAULT: 1

For compositional runs, this item refers to an EoS region number. In such a case, the gas properties are calculated using the equation of state for the given region.

6. Temperature in aquifer
   This item is ignored unless either the thermal or temperature option has been activated (see keywords THERMAL and TEMP).
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

Each record should be terminated with a slash (/). The set of records should be terminated with a blank record containing just a slash.

Note: The slash could be omitted for ECLIPSE 100, especially if the number of records entered using the AQUCHGAS keyword is the same as the number specified in AQUUDIMS. If not, a warning message regarding “spurious data” will be output, but the run will carry on as before.

Connections to a constant head/pressure gas ‘aquifer’ defined by this keyword are read in by either the AQUANCON or AQUANCONL keyword, which allows more than one face of the reservoir to be connected to an aquifer.

The aquifer properties can be modified at any time during the simulation by reentering the AQUCHGAS keyword in the SCHEDULE section.

Keywords
**ECLIPSE Reference Manual**

**ECLIPSE 300 only**

**Note:** For compositional runs, the **AQSTREAM** keyword must be used together with the **AUCHGAS** keyword.

The aquifer inflow is given by the equation:

\[ Q_{ai} = \frac{d}{dt} (G_{ai}) = J\alpha_i [p_a - p_c - p_i + \rho g (d_i - d_a)] \]

Eq. 3.3

where

- \( Q_{ai} \) is the inflow rate from the aquifer to the connecting grid block i
- \( G_{ai} \) is the cumulative influx from the aquifer to grid block i
- \( J \) is the specified Productivity Index of the aquifer
- \( \alpha_i \) is the area fraction for the connection to grid block i
- \( p_a \) is the pressure in the aquifer
- \( p_i \) is the gas pressure in a connecting grid block i
- \( \rho \) is gas density in the aquifer
- \( d_i \) is the grid block depth
- \( d_a \) is the datum depth of the aquifer.
- \( p_c \) is the capillary pressure

**Example**

One constant head gas 'aquifer', connected to two vertical sides of a 10 x 10 x 2 grid:

```
AUCHGAS
  1 0.0 1.0 2.0E9 1 /
/
AQUANCON
  1 1 10 10 1 2 'J+' /
  1 10 10 1 10 1 2 'I+' /
```

Keywords

348
AQUCHWAT  Specifies the property data for constant head/pressure water aquifers

The number of analytic aquifers is specified using the keyword in the RUNSPEC section.

Each record specifies the properties of a separate constant head aquifer (ranging from 1 to the number of analytic aquifers), and contains the following items of data:

1. Aquifer identification number.
   This should be a number between 1 and the maximum number of aquifers, entered using the keyword AQUDIMS.

2. Datum depth
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

3. Describes the input to item 4
   PRESSURE
   HEAD
   DEFAULT: PRESSURE

4. Pressure in the aquifer at the datum depth if item 3 is PRESSURE
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
   or
   Head in the aquifer at the datum depth if item 3 is HEAD
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

   If this quantity is defaulted or given a negative value, the initial pressure is calculated to place the aquifer as nearly as possible in equilibrium with the reservoir.

   Note: If using the head option the head, H, is immediately converted by ECLIPSE to a pressure using the formula $p_a = \rho g (H - d_a) + 1$ atmosphere. ECLIPSE 300 calculates the density, $\rho$, from the table given in item 6. ECLIPSE 100 uses the water density specified with the DENSITY keyword.

5. Aquifer productivity index
   (total influx rate per unit pressure difference)
   UNITS: sm$^3$/day/bars (METRIC), stb/day/psi (FIELD), scc/hr/atm (LAB), sm$^3$/day/atm (PVT-M)

6. Table number for water pressure properties (see keyword PVTW).
   DEFAULT: 1

ECLIPSE 300

For the GASWAT, CO2STORE, CO2SOL, GASSOL or COMPW options this item refers to an EoS region number. In such a case, the water properties are calculated using the equation of state for the given region for GASWAT or CO2STORE and the water properties data for COMPW or CO2SOL or GASSOL. If item 119 of OPTIONS3 is set to 1, this reverts to a table number.

7. Initial salt concentration in the aquifer
This item is ignored unless either the Brine option, or the Salt-sensitivity option in the Polymer Flood Model, has been activated (keyword BRINE in RUNSPEC).

UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scce (LAB), kg/sm$^3$ (PVT-M)
DEFAULT: 0.0

Note: If items 8 and 9 are defined, if one is defined they must both be defined, then the pressure, $p_i$, in the formula for the flux is replaced with $min (max (p_{min}, p_{i}), p_{max})$. If heads are being used ECLIPSE immediately converts them to the appropriate pressure using the formula given above.

8. Minimum pressure if item 3 is PRESSURE
UNITS: barsa (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)
DEFAULT: -infinity
or
Minimum head if item 3 is HEAD
UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
DEFAULT: -infinity

9. Maximum pressure if item 3 is PRESSURE
UNITS: barsa (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)
DEFAULT: infinity
or
Maximum head if item 3 is HEAD
UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
DEFAULT: infinity

10. Do you want to ignore the capillary pressure term in the equation 3.4? (The alternative way of regarding this question is, do you want the aquifer pressure given in item 4 to account for all pressure effects?) YES or NO?
DEFAULT: ‘NO’

11. Minimum allowed flow per aquifer connection
UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)
DEFAULT: -infinity

12. Maximum allowed flow per aquifer connection
UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)
DEFAULT: infinity

13. Do you want to remove the depth term, $J_0 \cdot \rho g (d_i - d_o)$, in equation 3.4? ‘YES’ or ‘NO’
DEFAULT: ‘NO’
14. Do you want to impose minimum and maximum flow rates (items 11 and 12) before or after the flow rate has been scaled by the grid block saturation in cases when water is flowing into aquifer from grid. This number should be zero if before and non-zero if after.

DEFAULT: 0

15. Temperature in aquifer

This item is ignored unless either the thermal or temperature option has been activated (see keywords THERMAL and TEMP).

UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

Each record should be terminated with a slash (/). The set of records should be terminated with a blank record containing just a slash (/).

Note: The slash could be omitted for ECLIPSE 100, especially if the number of records entered using the AQUCHWAT keyword is the same as the number specified in AQUDIMS. If not, a warning message regarding “spurious data” will be output, but the run will carry on as before.

Connections to a constant head aquifer defined by this keyword are read in by either the AQUANCON or the AQANCONL keyword, which allows more than one face of the reservoir to be connected to an aquifer.

The aquifer properties can be modified at any time during the simulation by entering the AQUCHWAT keyword again in the SCHEDULE section.

Note: For the GASWAT and CO2STORE options, the AQSTREAM keyword may be used to give the composition of a specified aquifer. By default, the aquifer contains only the component “H2O”. Any water component is allowed to leave the reservoir or enter it, depending on the aquifer composition. If item 131 of OPTIONS3 is set to 1, this reverts to the pre-2007.2 behavior where the water entering / leaving the reservoir is made up entirely of the “H2O” component.

For the multi-component water option (COMPW), the AQSTREAM keyword may be used to give the composition of a specified aquifer. By default, the aquifer contains only the last water component. Any water component is allowed to leave the reservoir or enter it, depending on the aquifer composition. If item 131 of OPTIONS3 is set to 1, this reverts to the pre-2008.1 behavior where the water entering / leaving the reservoir is made up entirely of the last water component.

For the CO2SOL and GASSOL options, any water aquifer is made up entirely of the “WATER” component. Any water component is allowed to leave the reservoir. If item 131 of OPTIONS3 is set to 1, this reverts to the pre-2008.1 behavior where the water entering / leaving the reservoir is made up entirely of the “WATER” component.

The aquifer inflow is given by the equation:

\[ Q_{ai} = \frac{d}{dt} (G_{ai}) = J a_i \left[ p_a + p_c - p_i + \rho g (d_i - d_a) \right] \]  

Eq. 3.4

where

- \( Q_{ai} \) is the inflow rate from the aquifer to the connecting grid block i
- \( G_{ai} \) is the cumulative influx from the aquifer to grid block i
is the specified productivity index of the aquifer

\( \alpha_i \) is the area fraction for the connection to grid block \( i \)

\( p_a \) is the pressure in the aquifer

\( p_i \) is the pressure in a connecting grid block \( i \)

\( \rho \) is water density in the aquifer

\( d_i \) is the grid block depth

\( d_a \) is the datum depth of the aquifer.

\( p_c \) is the capillary pressure.

**Example**

One constant head aquifer, connected to two vertical sides of a 10 x 10 x 2 grid:

```
AUCHWAT
  1 0.0 'PRESSURE' 1.0 2.0E9 1 /
/
AUCANCON
  1 1 10 10 10 1 2 'J+' /
  1 10 10 1 10 1 2 'I+' /
/
```

Keywords

352
**AQUCON**

**Specifies connection data for numerical aquifers**

AQUCON connects a numerical aquifer (declared using the **AQNUM** keyword) to one or more reservoir cells. The connection to the reservoir is set up by an arbitrary box defined by lower and upper I, J and K indices.

For each active grid block within the box, an aquifer connection is made to the face specified in item 8 if either of the following conditions are met:

- The face is on the outside of the reservoir, or
- The face adjoins an inactive cell.

**Note:** Care must be taken while considering the IJK box extents. Multiple thin layers of cells on the reservoir boundary are strongly recommended over bulky boxes covering inner regions. ECLIPSE does not check for aquifer connections inside the reservoir. As a result, any cell within the prescribed box may be attached an aquifer as long as its adjacent cell is inactive. There are various situations when cells may be inactive inside the reservoir, for example due to their small pore volume (see **MINPV**) or when a sector model (see **USEFLUX**) is surrounded by inactive cells of the full-field model.

Any number of records may be entered, each terminated with a slash (/). The set of records must be terminated with a blank record containing just a slash.

The total number of lines of AQUCON data must be less than or equal to **MXNAQC** set in item 2 of keyword **AQUDIMS** in the RUNSPEC section.

**ECLIPSE 100**

The keyword can be specified in the **GRID** section.

**ECLIPSE 300**

The keyword can be specified in the **GRID** and **SOLUTION** sections.

The items in each record are:

1. Aquifer identification number
   
   This should be a number between 1 and the maximum number of aquifers, set in the keyword **AQUDIMS**.

2. Lower I-coordinate of cells to be connected
3. Upper I-coordinate of cells to be connected
4. Lower J-coordinate of cells to be connected
5. Upper J-coordinate of cells to be connected
6. Lower K-coordinate of cells to be connected
7. Upper K-coordinate of cells to be connected
8. Face of the cell to be connected to the aquifer.

   This should be one of I+, I-, J+, J-, K+ or K-, where K+ for instance means the bottom face. Alternatively, X+, X-, Y+, Y-, Z+ or Z- could be chosen where Z+ for instance means the bottom face.

9. The transmissibility multiplier
   
   The multiplier enables the calculated transmissibility value to be multiplied by a user defined factor, for example in history matching.
10. Transmissibility option

The option enables the aquifer connection transmissibility to be calculated based on either the aquifer cross-sectional area (specified in item 5 of AQNUM) or the aquifer connection cell area. A value of 0 uses the cross-sectional area specified in item 5 of AQNUM, whereas a value of 1 uses the cross-sectional area calculated from the aquifer connection cell. Note: the second option is only used for non-radial grids.

An integer, either 0 or 1

DEFAULT: 0

11. Allow aquifer connections to cell faces adjoining active cells?

Either YES or NO

If this data item is set to YES, the restriction that a connected face must not adjoin an active cell is lifted, allowing aquifer connections into the middle of the reservoir. This option should be used with care and is not recommended for general use.

DEFAULT: NO

12. VE option only. VE fraction (VEFRAC) for relative permeability calculations for aquifer-to-grid connections.

A value of 1.0 gives pure VE relative permeabilities with the whole face available for flow; while a value of 0.0 gives pure rock curves.

DEFAULT: 1.0

13. VE option only. VE fraction (VEFRACP) for capillary pressure calculation for aquifer-to-grid connections.

A value of 1.0 gives pure VE pseudo capillary pressures, while a value of 0.0 gives pure rock curves.

DEFAULT: 1.0

The transmissibility between the numerical aquifer and grid block is calculated as follows:

\[ T = \frac{\text{CDARCY}}{(1/T_a) + (1/T_c)} \]  

Eq. 3.5

where

\[ T_c = \text{Component of transmissibility from the grid block} \]

\[ T_a = \text{Component of transmissibility from the aquifer} \]

In the Cartesian case:

\[ T_c = \frac{K_c \cdot A_c \cdot \text{NTG}}{D_c} \]  

Eq. 3.6

where

\[ K_c = \text{Cell permeability in the appropriate direction} \]
\( A_c \) = Cell face area

\( NTG \) = The cell net-to-gross ratio if the cell face X or Y

\( D_c \) = Distance from the cell center the face

If transmissibility option 0 (item 10) is used:

\[
T_a = \frac{K_a \cdot A_a}{D_a}
\]

Eq. 3.7

where

\( K_a \) = Aquifer permeability from \( \text{AQUNUM} \)

\( A_a \) = Aquifer area from \( \text{AQUNUM} \)

\( D_a \) = Half the aquifer length from \( \text{AQUNUM} \)

If transmissibility option 1 (item 10) is used:

\[
T_a = \frac{K_a \cdot A_c}{D_a}
\]

Eq. 3.8

**ECLIPSE 100**

The NNCs that both connect the aquifer cells and connect the aquifer to the reservoir can be listed by using the \text{ALLNNC} mnemonic (24th control) in the \text{RPTGRID} keyword.

**ECLIPSE 300**

For this simulator, the NNCs can be output using the \text{NNC} mnemonic in the \text{RPTGRID} keyword.

In addition, an array with the name \text{AQUIFERN} is written to the \text{INIT} file to illustrate which cells are connected to each aquifer. If a cell is connected to aquifer \( N \), then the number \( 2^{(N-1)} \) will appear in the \text{AQUIFERN} array. If a cell is connected to multiple aquifers, then the appropriate output numbers are summed. Cells representing the aquifer itself are given a negative number.

See also "Aquifer modeling facilities" in the \textit{ECLIPSE Technical Description}.

**Examples**

**Example 1**

An example is the connection of aquifer number 2 to all the cells in the first three J-K planes with a free face in the I-direction. This attaches the aquifer to the reservoir face, even if there are inactive cells between the grid edge and the reservoir. It is assumed the grid has \( \text{NX}=20, \text{NY}=20, \text{NZ}=10 \).

\begin{verbatim}
AQUICON
 2 1 3 1 20 1 10 'I-' 1.0 /
\end{verbatim}

**Example 2**

In the following, aquifer 1 is connected to cells in the first three layers of the fifth column of an X-Z cross-section. The I+ face is used, so connections are only made if the corresponding cells of column 6 are inactive, or if \text{NDIVIX} (specified in RUNSPEC keyword \text{DIMENS}) is 5.
AQUCON 1 5 5 1 1 3 'I+' 1.0 /
AQUCT

Specifies the property data for Carter-Tracy aquifers

This keyword specifies Carter-Tracy aquifers.

In ECLIPSE 100, the keyword can be specified in the SOLUTION and SCHEDULE sections.

In ECLIPSE 300, the keyword can be specified in the GRID, PROPS, SOLUTION and SCHEDULE sections.

The number of analytic aquifers is entered using the AQUDIMS keyword in the RUNSPEC section.

Each record refers to a separate Carter-Tracy aquifer and contains the following items of data:

1. Aquifer identification number
   This should be a number between 1 and the maximum number of aquifers, entered using the AQUDIMS keyword.

2. Datum depth
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

3. Initial aquifer pressure at the datum depth
   If this quantity is defaulted, the initial pressure is calculated to place the aquifer as nearly as possible in equilibrium with the reservoir.
   UNITS: barsa (METRIC), psia (FIELD), ata (LAB), ata (PVT-M)

4. Permeability of the aquifer
   UNITS: mD (METRIC), mD (FIELD), mD (LAB), mD (PVT-M)

5. Porosity of the aquifer
   UNITS: m$^3$/m$^3$ (METRIC), ft$^3$/ft$^3$ (FIELD), cc/cc (LAB), m$^3$m$^3$ (PVT-M)
   DEFAULT: 1.0

6. Total (rock + water) compressibility of the aquifer
   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/ata (LAB), 1/ata (PVT-M)

7. External radius of reservoir or the inner radius of the aquifer
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

8. Thickness of the aquifer
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

9. Angle of influence (the angle subtended by the boundary between the reservoir and the aquifer)
   UNITS: degrees (METRIC), degrees (FIELD), degrees (LAB), degrees (PVT-M)
   DEFAULT: 360.0 degrees

10. Table number for water pressure properties
    (see keyword WATERTAB (ECLIPSE 300 only), or PVTW)
For the **GASWAT**, **CO2STORE**, **CO2SOL**, **GASSOL** or **COMPW** options this item refers to an EoS region number. In such a case, the water properties are calculated using the equation of state for the given region for **GASWAT** or **CO2STORE** and the water properties data for **COMPW** or **CO2SOL** or **GASSOL**. Note that, for **CO2SOL**, the “viscosibility” term of the first water PVT table defined using PVTW is used during the calculation of the aquifer water viscosity. If item 119 of **OPTIONS3** is set to 1, this reverts to a table number.

**DEFAULT**: Table 1

11. Table number for influence functions, which are the dimensionless time and pressure values (see the **AQUTAB** keyword)

**DEFAULT**: Table 1

Table 1 is the constant terminal rate case for an infinite aquifer as given by Van Everdingen and Hurst. This table is automatically supplied by the simulator.

12. Initial salt concentration in the aquifer

This item is ignored unless either the Brine option, or the Salt-sensitivity option in the Polymer Flood Model, has been activated (keyword **BRINE** in **RUNSPEC**).

**UNITS**: kg/sm³ (METRIC), lb/stb (FIELD), gm/scf (LAB), kg/sm³ (PVT-M)

**DEFAULT**: 0.0

13. Temperature in aquifer

This item is ignored unless either the thermal or temperature option has been activated (see keywords **THERMAL** and **TEMP**).

**UNITS**: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

Each record is terminated with a slash (/). The set of records should be terminated with a blank record containing just a slash.

**ECLIPSE 100**

*Note*: The slash could be omitted for ECLIPSE 100, especially if the number of records entered using the **AQUCT** keyword is the same as the number specified in **AQUUDIMS**. If not, a warning message regarding “spurious data” will be output, but the run will carry on as before.

Connections to a Carter-Tracy aquifer defined by this keyword are read in by either the **AQUANCON** or the **AQANCONL** keyword, which allows more than one face of the reservoir to be connected to a Carter-Tracy aquifer.

The aquifer properties can be modified at any time during the simulation by entering the **AQUCT** keyword again in the **SCHEDULE** section.

**ECLIPSE 300**

*Note*: For the **GASWAT** and **CO2STORE** options, the **AQSTREAM** keyword may be used to give the composition of a specified aquifer. By default, the aquifer contains only the component “H2O”. Any water component is allowed to leave the reservoir or enter it, depending on the aquifer composition. If item 131 of **OPTIONS3** is set to 1, this reverts to the pre-2007.2 behavior where the water entering / leaving the reservoir is made up entirely of the “H2O” component.

For the multi-component water option (**COMPW**), the **AQSTREAW** keyword may be used to give the composition of a specified aquifer. By default, the aquifer contains only the last water component. Any
water component is allowed to leave the reservoir or enter it, depending on the aquifer composition. If item 131 of OPTIONS3 is set to 1, this reverts to the pre-2008.1 behavior where the water entering / leaving the reservoir is made up entirely of the last water component.

For the CO2SOL and GASSOL options, any water aquifer is made up entirely of the “WATER” component. Any water component is allowed to leave the reservoir. If item 131 of OPTIONS3 is set to 1, this reverts to the pre-2008.1 behavior where the water entering / leaving the reservoir is made up entirely of the “WATER” component.

The AQUCT keyword cannot be used with the keyword AQUFET. It is also possible to use the AQUCT keyword in the GRID and PROPS sections. See also "Aquifer modeling facilities" of the ECLIPSE Technical Description.

**Note:** AQUCT must not be used in the SOLUTION section of restarted runs, as the aquifer specification data cannot be changed.

**Example**

One Carter-Tracy aquifer, connected to two vertical sides of a 10 x 10 x 2 grid:

```plaintext
AQUCT
  1 7000.0 4000.0 200.0 .3 1.0E-5 500.0 50.0 10.0 1 1 /
/
AQUANCON
  1 1 10 10 10 1 2 'J+' /
  1 10 10 1 10 1 2 'I+' /
/
```

**Keywords**

359
AUCWFAC  Changes the property data for all constant head aquifers

This keyword can be used to change the datum depth and the datum pressures in all the aquifers defined using the AUCHWAT keyword.

The data consists of some or all of the following items terminated with a slash (/).

1. Change to add to datum depths
   
   **UNITS**: m (METRIC), ft (FIELD), cm (LAB)
   
   **DEFAULT**: 0.0

2. Factor to multiply datum pressures by
   
   **DEFAULT**: 1.0

**Example**

Increase the datum depth by 10 metres; leave the pressure unchanged:

```
AUCWFAC
10.0 1.0 /
```
Dimensions for aquifers

The data consists of some or all of the following items which describe the aquifers employed during the run (see "Aquifer modeling facilities" in the ECLIPSE Technical Description). The data must be terminated by a slash (/).

1. **MXNAQN** The maximum number of lines of numerical aquifer data entered in keyword **AQUNUM** in the GRID section.
   
   DEFAULT: 1

2. **MXNAQC** The maximum number of lines of connection data for numerical aquifers entered in keyword **AQUCON** in the GRID section.
   
   DEFAULT: 1

3. **NIFTBL** The maximum number of influence tables for Carter-Tracy aquifers, entered by the keyword **AQUTAB** in the PROPS section. This should be greater than 1 if there are any user supplied tables, as table number 1 is a built-in default table.
   
   DEFAULT: 1

4. **NRIFTB** The maximum number of rows in a Carter-Tracy aquifer influence table. As table 1 contains 36 rows, a minimum value of 36 is always recorded.
   
   DEFAULT: 36

5. **NANAQU** The maximum number of analytic aquifers in the model.
   
   DEFAULT: 1

6. **NCAMAX**

   **ECLIPSE 100**
   
   The maximum number of grid blocks connected to any single analytic aquifer.

   **ECLIPSE 300**
   
   The maximum number of lines of analytic aquifer connection data.
   
   DEFAULT: 1

   **ECLIPSE 300**

   Note: No distinction is made between MXNAQN, the maximum number of lines in the numerical aquifers, and NANAQU, the maximum number of analytic aquifers. This is the same between MXNAQC and NCAMAX, the number of connections to numerical and analytic aquifers. If two values are input then the maximum is used. Although there is a difference in the specification of item 6 of this keyword between the two simulators, ECLIPSE 300 will internally determine the maximum number of connections for any aquifer.

7. **MXNALI** The maximum number of aquifer lists.
   
   DEFAULT: 0

8. **MXAAQL** The maximum number of analytic aquifers in any single aquifer list.
   
   DEFAULT: 0

   **Note:** Any aquifer identification number used in aquifer keywords must lie between 1 and the sum of MXNAQN and NANAQU.
Example

Two analytic aquifers, up to 100 grid block connections each. Default influence table.

```
AQUIDIMS
 0 0 1 36 2 100 /
```
AQUFET

Specification data for Fetkovich aquifers

This keyword can be used to supply both the aquifer properties and the cell connections to Fetkovich aquifers.

**Note:** This keyword can only be used in ECLIPSE 100.

The connection to the reservoir is set up by an arbitrary box defined by lower and upper I, J and K indices (refer to items 7-12 below). It allows only one reservoir face to be connected to any single aquifer. If an aquifer is required to connect to two or more faces, use the keywords AQUFETP and AQUANCON or AQANCONL, instead of AQUFET.

The number of analytic aquifers is specified using the AQUUDIMS keyword.

Each record refers to a separate Fetkovich aquifer (ranging from 1 to the number of analytic aquifers), and contains the following items of data:

1. Datum depth
   
   **UNITS:** m (METRIC), ft (FIELD), cm (LAB)

2. Initial aquifer pressure at the datum depth
   
   If this quantity is defaulted or given a negative value, the initial pressure is calculated to place the aquifer as nearly as possible in equilibrium with the reservoir.
   
   **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB)

3. Initial volume of water in the aquifer
   
   **UNITS:** sm$^3$ (METRIC), stb (FIELD), scc (LAB)

4. Total (rock + water) compressibility of the aquifer
   
   **UNITS:** 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB)

5. Aquifer productivity index
   
   (total influx rate per unit pressure difference)
   
   **UNITS:** sm$^3$/day/bars (METRIC), stb/day/psi (FIELD), scc/hr/atm (LAB)

6. Table number for water pressure properties
   
   (See keyword PVTW)

7. Lower I-location of connecting grid blocks

8. Upper I-location of connecting grid blocks

9. Lower J-location of connecting grid blocks

10. Upper J-location of connecting grid blocks

11. Lower K-location of connecting grid blocks

12. Upper K-location of connecting grid blocks

13. Index defining the face of the reservoir to which the aquifer connects (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS)
14. Initial salt concentration in the aquifer

This item is ignored unless either the brine option, or the salt-sensitivity option in the polymer flood model, has been activated (see the BRINE keyword in the RUNSPEC section).

UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/sec (LAB)

DEFAULT: 0.0

Each record should be terminated with a slash (/).

The aquifer connects with all the grid blocks that lie within the specified bounds and have no active neighboring block on the side facing the aquifer. The total number of connecting grid blocks must not exceed the value entered using item 6 in keyword AQUEDIMS in the RUNSPEC section.

The aquifer productivity index is apportioned between the connecting grid blocks in proportion to the area of their connecting faces.

This keyword AQUFET cannot be used together with the keywords AQUFEPT, AQUCT, AQUANCON or AQUANCONL.

This keyword must not be used in restarted runs, as the aquifer specification data cannot be changed.

See also "Aquifer modeling facilities" in the ECLIPSE Technical Description for further information.

Example

One aquifer, on the bottom of a 10 x 10 x 5 grid:

```
AQUFET
7000 4000 2.0E9 1.0E-5 500.0 1 1 10 1 10 5 5 'K+' /
```
AQUFETP  Specifies the property data for Fetkovich aquifers

The number of analytic aquifers is specified using the AQUDIMS keyword in the RUNSPEC section. Each record specifies the properties of a separate Fetkovich aquifer (ranging from 1 to the number of analytic aquifers), and contains the following items of data:

1. Aquifer identification number
   This should be a number between 1 and the maximum number of aquifers, entered using the keyword AQUDIMS.

2. Datum depth
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

3. Initial aquifer pressure at the datum depth
   If this quantity is defaulted, the initial pressure is calculated to place the aquifer as nearly as possible in equilibrium with the reservoir.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

4. Initial volume of water in the aquifer
   UNITS: sm$^3$ (METRIC), stb (FIELD), scc (LAB), sm$^3$ (PVT-M)

5. Total (rock + water) compressibility of the aquifer
   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

6. Aquifer productivity index
   (total influx rate per unit pressure difference)
   UNITS: sm$^3$/day/bars (METRIC), stb/day/psi (FIELD), scc/hr/atm (LAB), sm$^3$/day/atm (PVT-M)

7. Table number for water pressure properties
   (see keyword PVTW).
   DEFAULT: 1

   For the GASWAT, CO2STORE, CO2SOL, GASSOL or COMPW options this item refers to an EoS region number. In such a case, the water properties are calculated using the equation of state for the given region for GASWAT or CO2STORE and the water properties data for COMPW or CO2SOL, or GASSOL. If item 119 of OPTIONS3 is set to 1, this reverts to a table number.
   DEFAULT: 1

8. Initial salt concentration in the aquifer
   This item is ignored unless either the Brine option, or the Salt-sensitivity option in the Polymer Flood Model, has been activated (keyword BRINE in RUNSPEC).
   UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm$^3$ (PVT-M)
   DEFAULT: 0.0

9. Temperature in aquifer
This item is ignored unless either the thermal or temperature option has been activated (see keywords \texttt{THERMAL} and \texttt{TEMP}).

\textbf{UNITS}: °C (\texttt{METRIC}), °F (\texttt{FIELD}), °C (\texttt{LAB}), °C (\texttt{PVT-M})

Each record should be terminated with a slash (/). The set of records should be terminated with a blank record containing just a slash (/).

\textbf{ECLIPSE 100}

\textit{Note}: The slash could be omitted for \texttt{ECLIPSE 100}, especially if the number of records entered using the \texttt{AQUFETP} keyword is the same as the \texttt{NANAQU} number specified in \texttt{AQUUDIMS}. If not, a warning message regarding “spurious data” will be output, but the run will carry on as before.

Connections to a Fetkovich aquifer defined by this keyword are read in by either the \texttt{AQUANCON} or the \texttt{AQANCONL} keyword, which allows more than one face of the reservoir to be connected to a Fetkovich aquifer.

\textbf{ECLIPSE 100}

This keyword \texttt{AQUFETP} cannot be used together with the keyword \texttt{AQUFET}.

The aquifer properties can be modified at any time during the simulation by entering the \texttt{AQUFETP} keyword again in the \texttt{SCHEDULE} section.

\textbf{ECLIPSE 300 only}

\textit{Note}: For the \texttt{GASWAT} and \texttt{CO2STORE} options, the \texttt{AQSTREAM} keyword may be used to give the composition of a specified aquifer. By default, the aquifer contains only the component “H2O”. Any water component is allowed to leave the reservoir or enter it, depending on the aquifer composition. If item 131 of \texttt{OPTIONS3} is set to 1, this reverts to the pre-2007.2 behavior where the water entering or leaving the reservoir is made up entirely of the “H2O” component.

For the multi-component water option (\texttt{COMPW}), the \texttt{AQSTREAM} keyword may be used to give the composition of a specified aquifer. By default, the aquifer contains only the last water component. Any water component is allowed to leave the reservoir or enter it, depending on the aquifer composition. If item 131 of \texttt{OPTIONS3} is set to 1, this reverts to the pre-2008.1 behavior where the water entering / leaving the reservoir is made up entirely of the last water component.

For the \texttt{CO2SOL} and \texttt{GASSOL} options, any water aquifer is made up entirely of the “WATER” component. Any water component is allowed to leave the reservoir. If item 131 of \texttt{OPTIONS3} is set to 1, this reverts to the pre-2008.1 behavior where the water entering / leaving the reservoir is made up entirely of the “WATER” component.

See also "Aquifer modeling facilities" in the \textit{ECLIPSE Technical Description} for further information.

\textit{Note}: \texttt{AQUFETP} must not be used in the \texttt{SOLUTION} section of restarted runs, as the aquifer specification data cannot be changed.

\textbf{Example}

One Fetkovich aquifer, connected to two vertical sides of a 10 x 10 x 2 grid:

\begin{verbatim}
AQUFETP
  1 7000.0 4000.0 2.0E9 1.0E-5 500.0 1 /
AQUANCON
  1 1 10 10 10 1 2 'J+' /
\end{verbatim}

\textbf{Keywords}
1 10 10 1 10 1 2 'I+' /
AQUFLUX  

Specifies a constant flux aquifer

The keyword should be followed by up to NANAQU records of data, each record terminated with a slash (/). NANAQU refers to the number of analytic aquifers, as entered under item 5 of the AQUDIMS keyword.

Each record specifies the properties of a separate aquifer (1 to NANAQU) and contains the following items of data:

1. Aquifer identification number
   This should be a number between 1 and the maximum number of aquifers, set in the keyword AQUDIMS.

2. Aquifer flux
   This is the water inflow rate per unit area of connected cell face.
   **UNITS:** sm$^3$/day/m$^2$ (METRIC), stb/day/ft$^2$ (FIELD), sccc/hr/cm$^2$ (LAB), sm$^3$/day/m$^2$ (PVT-M)

3. Salt concentration in the aquifer
   This item is ignored unless either the Brine option, or the salt sensitivity option in the Polymer Flood Model, has been activated (keyword BRINE in RUNSPEC).
   **UNITS:** kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scce (LAB), kg/sm$^3$ (PVT-M)
   **DEFAULT:** 0.0

4. Temperature in aquifer
   This item is ignored unless either the thermal or temperature option has been activated (see keywords THERMAL and TEMP).
   **UNITS:** °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

5. Pressure in aquifer at datum depth
   This pressure is used along with the temperature in the aquifer to calculate the enthalpy of the aquifer and is ignored unless either the thermal or temperature option has been activated (see keywords THERMAL and TEMP).
   **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atm (PVT-M)

The set of records should be terminated with a blank record containing just a slash (/).

**ECLIPSE 100 only**

**Note:** The slash could be omitted for ECLIPSE 100, especially if the number of records entered using the AQUFLUX keyword is the same as the NANAQU number specified in AQUDIMS. If not, a warning message regarding “spurious data” will be output, but the run will carry on as before.

The grid block connections to each constant flux aquifer defined by this keyword must be specified by either the AQUANCON or the AQUANCONL keyword.

**ECLIPSE 300 only**

**Note:** For the GASWAT and CO2STORE options, the AQSTREAM keyword may be used to give the composition of a specified aquifer. By default, the aquifer contains only the component “H2O”. Any water component is allowed to leave the reservoir or enter it, depending on the aquifer composition. If item 131
of OPTIONS3 is set to 1, this reverts to the pre-2007.2 behavior where the water entering / leaving the reservoir is made up entirely of the “H2O” component.

For the multi-component water option (COMPW), the AQSTREAM keyword may be used to give the composition of a specified aquifer. By default, the aquifer contains only the last water component. Any water component is allowed to leave the reservoir or enter it, depending on the aquifer composition. If item 131 of OPTIONS3 is set to 1, this reverts to the pre-2008.1 behavior where the water entering / leaving the reservoir is made up entirely of the last water component.

For the CO2SOL and GASSOL options, any water aquifer is made up entirely of the “WATER” component. Any water component is allowed to leave the reservoir. If item 131 of OPTIONS3 is set to 1, this reverts to the pre-2008.1 behavior where the water entering or leaving the reservoir is made up entirely of the “WATER” component.

The water flow rate into a grid block from a constant flux aquifer is given by:

\[ Q_w = F_w \cdot A \cdot M \]  

Eq. 3.9

where

\( F_w \) is the flux, entered in item 2

\( A \) is the area of the connected cell face

\( M \) is the aquifer influx multiplier (item 10 of the AQUANCON keyword or item 11 of the AQANCONL keyword), which defaults to 1.0.

The area is calculated directly from the connected cell geometry, and is not affected by item 9 of the AQUANCON keyword or item 10 in the AQANCONL keyword.

The aquifer flux can be modified at any time during the simulation by reentering the AQUFLUX keyword in the SCHEDULE section.

The keyword AQUFLUX cannot be used together with the keyword AQUFET.

See also "Aquifer modeling facilities" in the ECLIPSE Technical Description for further information.

Example

Two constant flux aquifers, connected to two vertical sides of a 10 x 10 x 4 grid

```plaintext
AQUFLUX
 1 0.0003 /
 2 0.0004 /
/

AQUANCON
  -- Aquifer 1
  1 1 10 10 1 4 'J+' /
  1 10 10 6 10 1 4 'I+' /
  -- Aquifer 2
  2 10 10 6 10 1 4 'I+' /
/
```

Keywords
**AQUNNC**

Set non-neighbor connection values explicitly for numerical aquifers

This keyword enables a specified transmissibility to be set up between grid cells and numerical aquifer cells which are defined using the AQUNUM keyword. Normally ECLIPSE will calculate the numerical aquifer connections from the data given in the AQUCON keyword, but the AQUNNC keyword allows for more generality in defining the connections.

The keyword consists of a line of data for each required aquifer non-neighbor connection, specifying the cells to be linked and the required transmissibility value. The cell coordinates are specified in natural order. Each line of data must be terminated by a slash (/). After the last non-neighbor connection is defined, an additional slash on a new line terminates the list.

The values in each data record are:

1. Aquifer number
2. I-index of first cell joined to the non-neighbor connection (IX)
3. J-index of first cell joined to the non-neighbor connection (IY)
4. K-index of first cell joined to the non-neighbor connection (IZ)
5. I-index of second cell joined to the non-neighbor connection (JX)
6. J-index of second cell joined to the non-neighbor connection (JY)
7. K-index of second cell joined to the non-neighbor connection (JZ)
8. Transmissibility value of the non-neighbor connection (TRAN)
   
   **UNITS:** cP.rm³/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB)
   
   **DEFAULT:** 0.0
9. Saturation table number to be associated with flow from the first to the second cell (IST1)
10. Saturation table number to be associated with flow from the second to the first cell (IST2)
11. Pressure table number to be associated with flow from the first to the second cell (IPT1)
12. Pressure table number to be associated with flow from the second to the first cell (IPT2)
13. The face associated with flow from the first to the second cell (ZF1)

   Choose either: X+, X-, Y+, Y-, Z+, Z-

   This information is used only for the VE option.

14. The face associated with flow from the second to the first cell (ZF2)

   Choose either: X+, X-, Y+, Y-, Z+, Z-

   This information is used only for the VE option.
15. Diffusivity value of the non-neighbor connection (DIFF)

   This item will be ignored unless the Molecular Diffusion option is activated (keyword DIFFUSE in RUNSPEC section).
UNITS: m (METRIC), ft (FIELD), cm (LAB)

DEFAULT: 0.0

Each line of data must be terminated by a slash (/). After the last non-neighbor connection is defined, an additional slash on a new line terminates the list. It is again possible to specify this keyword more than once in the grid data.

The last seven items are optional. Therefore, if the slash is placed after the TRAN item, the table numbers is defaulted to those for the cells joined.

Example

```
AQUUNNC
  -- IA IX IY IZ JX JY JZ TRAN
  1  2  3  5  5  7  5  2.0 /
  1  2  3  6  8  6  5  4.5 /
/```

---
AQNUM  Assigns a numerical aquifer to a block

This keyword specifies blocks representing one-dimensional numerical aquifers. One or more aquifers may be set up. Each line of AQNUM declares that a given grid block is to represent a numerical aquifer. This enables its properties (such as initial pressure and volume) to be independent of its size and position in the grid. To complete the description of the aquifer, connections to other blocks may be made using the AQUCON keyword.

The total number of lines of data following the AQNUM keyword must be less than or equal to the value set by the 1st item in the AQUIMS keyword in the RUNSPEC section.

Each record must be terminated by a slash (/). The set of records must be terminated with a blank record containing just a slash.

The items in AQNUM are as follows:

1. Aquifer identification number
   This should be a number between 1 and the maximum number of aquifers, set in the keyword AQUIMS.

2. I-coordinate of grid block to represent aquifer

3. J-coordinate of grid block to represent aquifer

   For dual porosity cases, this should be in the fracture. Additionally, for dual porosity (single permeability) cases, the inactive grid block with these I,J,K coordinates should be set as dual porosity if the DPNUM keyword is used. It is possible to use DPNUM to set the inactive grid block as single porosity and then place the K-coordinate in the matrix, but this is not recommended as it does not use DPNUM default values.

5. Cross-sectional area
   This may be larger than can be accommodated within the grid block I,J,K. However, the position of the grid block on graphical displays is unchanged.
   UNITS: m² (METRIC), ft² (FIELD), cm² (LAB), m² (PVT-M)

6. Length
   This may be larger than can be accommodated within the grid block I,J,K.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

7. Porosity
   UNITS: m³/m³ (METRIC), ft³/ft³ (FIELD), cc/cc (LAB), m³/m³ (PVT-M)
   DEFAULT: Grid cell value

8. Permeability
   UNITS: mD (METRIC), mD (FIELD), mD (LAB), mD (PVT-M)

9. Aquifer depth
   This may differ from the geometric position in the reservoir. It may be defaulted, in which case the grid block depth is obtained.
10. Initial pressure

This may be specified explicitly or defaulted. If defaulted, the initial pressure is set by the equilibration procedure to be in hydrostatic equilibrium with the reservoir. The default procedure is recommended except in the case in which all initial pressures are entered by the user.

UNITs: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

11. PVT table number for aquifer

DEFAULT: Block PVTNUM value.

12. Saturation table number for aquifer

DEFAULT: Block value.

Notes

- The length, cross-sectional area and porosity are used to calculate the aquifer volume. The length, cross-sectional area and permeability are used to calculate the aquifer transmissibility.
- The table number used for flow into the aquifer will be that used for normal grid flows.
- In dual porosity runs the aquifer should be placed in the lower half (that is the fracture part) of the grid.
- The aquifer pore volume is calculated from the data entered in the AQUNUM keyword in all cases using

\[ \text{PORV} = \text{Porosity} \cdot \text{Length} \cdot \text{Area} \]  \hspace{1cm} \text{Eq. 3.10}

where the porosity is taken from the grid block value if defaulted. Any values of MULTPV entered in the GRID or EDIT sections does not apply to numerical aquifer cells. Any edits to PORV in the EDIT section for numerical aquifer cells are ignored and the aquifer pore volume is always calculated from equation 3.10. See also “Aquifer modeling facilities” in the ECLIPSE Technical Description.

- In parallel runs, defining a numerical aquifer in one grid partition and connecting it to cells in other partitions leads to the so-called off-processor connections which can degrade the overall performance. The number of off-processor connections is reported as part of the warning when this situation occurs. It is recommended that the creation of off-processor connections is avoided, or at least that their number is reduced. If this is not possible, try to increase NSTACK and LITMAX but expect more linear iterations per timestep. Note that analytical aquifers do not require any off-processor connections at all.

Example

In the following example a four cell aquifer is defined, with default porosity, depth and initial pressure. Aquifer-grid connections are made to the first cell specified with a given aquifer number.

```
AQUNUM
  1 7 1 1 1.0E+5 20000 1* 200.0 2* 1 1 /
  1 8 1 1 1.0E+5 20000 1* 200.0 2* 1 1 /
  1 9 1 1 1.0E+5 20000 1* 200.0 2* 1 1 /
  1 10 1 1 1.0E+5 20000 1* 200.0 2* 1 1 /
```

Keywords

373
Influence function tables for Carter-Tracy aquifers

The number of influence function tables is specified by the AQUDIMS keyword.

These tables are of dimensionless time and pressure, each terminated by a slash (/).

The first table is the default table that cannot be changed. Table one is defined and reserved for the constant terminal rate case for an infinite aquifer as given by Van Everdingen and Hurst.

The remaining tables entered using this keyword are numbered from 2 onwards.

Each table consists of 2 columns of data:

1. Dimensionless time
   Values should be greater than 0 and should increase monotonically down the column.
2. The corresponding dimensionless pressure
   Values should be greater than 0.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than the maximum value specified as the 4th item in the AQUDIMS keyword.

Note: The number of rows (specified in AQUDIMS) is always set to at least 36, to accommodate the first, default, table.

Example

Example with two influence function tables set in AQUDIMS. The first is the default table, and the second, is read in with AQUTAB.

```
AQUTAB
0.01 0.112
0.05 0.229
0.1  0.315
0.15 0.376
0.2  0.424
0.25 0.469
0.3  0.503
0.4  0.564
0.5  0.616
0.6  0.659
```

See "Aquifer modeling facilities" in the ECLIPSE Technical Description for further information.
Asphaltene deposition data

This keyword is set to provide data for the asphaltene deposition process. The required data consists of:

1. A static deposition or adsorption coefficient $\alpha$
   \[
   \text{UNITS: } 1/\text{day (METRIC)}, 1/\text{day (FIELD)}, 1/\text{hr (LAB)}, 1/\text{day (PVT-M)}
   \]

2. A plugging coefficient $\gamma_1$, used in the pre-2011.1 deposition model
   \[
   \text{UNITS: } 1/\text{m}^3 \text{ (METRIC)}, 1/\text{ft}^3 \text{ (FIELD)}, 1/\text{cc (LAB)}, 1/\text{m}^3 \text{ (PVT-M)}
   \]

3. An entrainment coefficient $\beta$
   \[
   \text{UNITS: } 1/\text{m} \text{ (METRIC)}, 1/\text{ft (FIELD)}, 1/\text{cm (LAB)}, 1/\text{m} \text{ (PVT-M)}
   \]

4. A critical velocity $U_{cr}$ above which entrainment takes place
   \[
   \text{UNITS: } \text{m/day (METRIC)}, \text{ ft/day (FIELD)}, \text{ cm/hr (LAB)}, \text{ m/day (PVT-M)}
   \]

5. A plugging coefficient $\gamma_2$ used in the three-component deposition model
   \[
   \text{UNITS: } 1/\text{m (METRIC)}, 1/\text{ft (FIELD)}, 1/\text{cm (LAB)}, 1/\text{m (PVT-M)}
   \]

All the above entries must be non-negative.

For more details please refer to the "Asphaltene option" in the ECLIPSE Technical Description.

Example

```
ASPDDEPO -- adsorp plug1 entrain Vcr plug2
0.001 0 1.e-5 0.01 0.1/
```
Asphaltene flocculation components

This keyword can be used to define the range of components that can precipitate as asphaltene, and the component number representing the flocs. Assuming that the list of components is enumerated as $1, 2, ..., i, i+1, ..., j, ..., k, ..., N$, this keyword should be followed by three integers corresponding to:

- the number of the first component that can precipitate, say $i$
- the number of the last component that can precipitate, say $j$
- the number of the component that represents the flocs, say $k$

This keyword has to be used in conjunction with the ASPFLRT keyword, which specifies the kinetic rates for the flocculation/dissociation processes.

Keyword CATYPE can be used instead of this keyword, and must be used if the Three component asphaltene deposition model is required.

For more details please refer to "Asphaltene option" in the ECLIPSE Technical Description.

Example

Run with 7 components, of which components 4 to 6 can precipitate and flocculate into component 7:

```plaintext
ASPFLOC
   -- cpfirst  cplast  cpflocs
   4         6         7 /
```
Asphaltene flocculation/dissociation rates

This keyword is used to provide the kinetic reaction rates for the flocculation of fines into flocs and the dissociation of flocs into fines. The rates must be positive numbers.

Assuming that there are $N$ components that can precipitate, then this keyword should be followed by $2N$ numbers corresponding to the rates.

- The first $N$ rates correspond to the flocculation process.
- The next $N$ rates correspond to the inverse process, whereby the flocs component (partially) dissociates.

This keyword has to be used in conjunction with the CATYPE or ASPFLRT keyword, which specifies the components that can precipitate and the flocs component number.

For more details please refer to "Asphaltene option" in the ECLIPSE Technical Description.

UNITS: 1/day (METRIC), 1/day (FIELD), 1/hr (LAB), 1/day (PVT-M)

Example

For a run where 3 components can precipitate and flocculate:

```
ASPFLRT -- flocc rates followed by dissociation rates
0.010  0.05  0.10
0.005  0.04  0.08 /
```
Asphaltene option

This keyword activates the asphaltene modeling option. The full asphaltene modeling facility is described in "Asphaltene option" in the ECLIPSE Technical Description.

This keyword contains data items that specify the asphaltene model options to be used.

1. A character string specifying the characterization criterion

   WEIGHT percentage molar weight of asphaltene dissolved in the oil as a function of specified variable(s). The data are then specified using either the set of keywords ASPP1P and ASPREWG for one variable, or the set of keywords ASPP2P and ASPPW2D for 2 variables.

   DEFAULT: WEIGHT

2. A character string specifying the permeability damage model (ASPKDAM keyword)

   PORO permeability damage are correlated to porosity changes (power-law model)

   TAB permeability damage is tabulated against asphaltene volume fraction deposit

   NO no permeability change

   DEFAULT: NO

   If permeability damage is to be modeled, the ASPKDAM keyword must be specified in the PROPS section with the corresponding data.

   This item is only used with the pre-2011.1 asphaltene deposition model. If the solid model is used for asphaltene deposition then this item is ignored. Permeability damage can be modeled using the SOLIDMMS keyword.

3. A character string specifying the viscosity model (ASPVISO keyword)

   EI Einstein model (highly diluted solution): one parameter

   KD Krieger-Dougherty model: two parameters

   TAB oil viscosity multiplier is tabulated against the asphaltene precipitation fraction

   NO no viscosity change

   DEFAULT: NO

   If oil viscosity change is to be modeled, the ASPVISO keyword must be specified in the PROPS section with the appropriate data.

Examples

Example 1

Asphaltene characterized by percentage weight of asphaltene component(s) in the oil, permeability damage defined by a solid saturation dependent mobility multiplier, and Krieger-Dougherty viscosity model.

```
ASPHALTE
-- charact. crit Perm damage Viscosity model
WEIGHT 1* KD/
```
Example 2

Asphaltenes default: weight, no permeability damage and no viscosity change:

```plaintext
ASPHALTE
-- charact. crit Perm damage Viscosity model
/
```
**ASPKDAM**

**Asphaltene permeability damage**

Setting this keyword provides data to model permeability damage due to asphaltene deposition. However, it can only be used with the pre-2011.1 asphaltene deposition model. If, instead of this model, the solid model is used for asphaltene deposition, then a solid dependent mobility multiplier should be used with keyword *SOLIDMMS* or *SOLIDMMC*.

There are two possible ways to model this damage:

1. by correlating the permeability change to the porosity change. This is done by setting the second item of the *ASPHALTE* keyword to ‘PORO’. Then this keyword provides the exponent in the power-law relationship between the permeability change and the porosity change.

2. by providing a look-up table giving the permeability multiplier as a function of the volume fraction of asphaltene deposit. This is done by setting the second item of the *ASPHALTE* keyword to ‘TAB’. In this case, the first column is the volume fraction of the asphaltene deposit and the second column is the permeability multiplier. The number of rows for this table can be set in item 17 of *TABDIMS*.

If the asphaltene volume fraction goes outside the specified range in the first column, the permeability multiplier is taken as the first or the last value specified in the table accordingly. No extrapolation is performed.

Depending on the model chosen, this keyword takes the appropriate number of data, as shown in the examples below. For more details please refer to “Asphaltene option” in the *ECLIPSE Technical Description*.

**Note:** The reported values using ASPKDM or BASKDM mnemonics should be interpreted as follows: a value of 1 means no damage whereas a value less than 1 means permeability damage.

### Examples

**Example 1**

Permeability change correlated to porosity changes

```
ASPKDAM
   -- exponent
       3 /
```

**Example 2**

Permeability change as a function of volume fraction of asphaltene deposit (see item 17 of *TABDIMS* for the maximum number of rows).

```
ASPKDAM
   -- deposit multiplier
       0.00  1.0
       0.01  0.9
       0.10  0.5 /
```
**ASPKROW**  Asphaltene relative permeability for wettability change

This keyword can be used to provide the oil wet relative permeability values for \( k_{rw} \) and \( k_{ro} \). The actual relative permeabilities are determined by interpolation between the oil wet values (provided by this keyword) and the water wet values (provided by standard keywords such as \texttt{SWFN}, \texttt{SOF3} or \texttt{SWOF}). Interpolation depends on the wettability factor \( F \) which is given by the \texttt{ASPWETF} keyword.

The keyword is followed by a number of tables, each containing three columns of data:

1. The water saturation.
2. The corresponding water relative permeability.
3. The corresponding oil relative permeability.

The number of tables to be provided with this keyword is equal to the number of saturation tables (see item 1 of the \texttt{TABDIMS} keyword), and the maximum number of rows in each table is set by item 3 of the \texttt{TABDIMS} keyword.

For more details please refer to the “Asphaltene option” in the ECLIPSE Technical Description.

*Note:* This keyword is active only where the system is water wet, defined here by the oil-water capillary pressure \( P_{cow} > 0 \)

### Example

```
ASPKROW
"-- Sw   krw    kro
  0.00  0.0    1.0
  0.50  0.31   0.7
  1.00  0.95   0.0/
```

---

**Keywords**

381
Asphaltene plugging control

This keyword can be used to control asphaltene plugging. The required data consist of:

1. A critical volume fraction of asphaltene deposit above which plugging can happen, either $S_{cr}$ or $\varepsilon_{cr}$.

   If the solid model is used for asphaltene deposition, the volume fraction is measured as a proportion of the pore volume, and the critical solid saturation $S_{cr}$ is required here. However, if the pre-2011.1 asphaltene deposition model is used, the volume fraction of asphaltene deposit is measured as a proportion of the cell volume and $\varepsilon_{cr} = \varphi \cdot S_{cr}$ is required here.

2. A critical asphaltene flocs molar concentration above which plugging can happen.

   UNITS: kg-M/rm$^3$ (METRIC), lb-M/rb (FIELD), gm-M/rcc (LAB), kg-M/rm$^3$ (PVT-M)

3. A character string equal to “AND” or “OR” that is used to combined the effects of the first two items.

   DEFAULT: “AND”

   The “OR” option is only used in the pre-2011.1 Asphaltene model. For the three-component (2011.1) model, plugging will only happen if both criteria are met. If a critical solid fraction is not required, set $S_{cr} = 0$.

All the numbers must be non-negative.

For more details please refer to the "Asphaltene option" in the ECLIPSE Technical Description.

Example

```
ASPLCRT
-- vol frac.  conc
  0.001   1.e-4   /
```
**Asphaltene one parameter precipitation**

This keyword provides data for the asphaltene precipitation in the case where the characterization is based on the percentage molar weight of specified component(s). The characterization is specified by the first item of the `ASPHALTE` keyword, and the component range is specified by the `ASPFLOC` keyword.

This keyword specifies the variable the percentage limit is set against. It can have the following data items. Valid items are:

- `P` (for pressure)
- `T` (for temperature)
- `Z` (for total molar fraction)

**DEFAULT: ‘P’**

If the first item is `Z` (for total molar fraction) then a second item must be provided to set the component number for the concentration.

This keyword should be used in conjunction with `ASPREWG` keyword to specify the percentage molar weight limit set against the variable specified here by the `ASPP1P` keyword.

For more details please refer to the "Asphaltene option" in the *ECLIPSE Technical Description*.

**Note:** The temperature can only be used in thermal runs.

### Examples

#### Example 1

The first item of the `ASPHALTE` keyword is `WEIGHT`, and precipitation set as a function of pressure:

```plaintext
ASPP1P
-- pressure
  'P' / 
```

#### Example 2

The first item of the `ASPHALTE` keyword is `WEIGHT`, and precipitation set as a function of the molar fraction of the second component:

```plaintext
ASPP1P
-- concentration 2nd component
  'Z' 2 / 
```
**Asphaltene two parameters precipitation**

This keyword provides data for the asphaltene precipitation in the case where the characterization is based on the percentage molar weight of specified component(s). The characterization is specified by the first item of the **ASPHALTE** keyword, and the component range is specified by the **ASPFLOC** keyword.

This keyword specifies the set of two variables that the percentage limit is set against. Valid items are:

- **P T** (for pressure and temperature)
- **P Z xx** (for pressure and total molar fraction of xx component)
- **T Z xx** (for temperature and total molar fraction of xx component)

If the total molar fraction is used as a variable, then a third item must be provided to set the component number for the concentration.

This keyword should be used in conjunction with the **ASPPW2D** keyword to specify the percentage molar weight limit set against the variable specified here by the **ASPP2P** keyword.

For more details please refer to the "Asphaltene option" in the *ECLIPSE Technical Description*.

**Note:** The temperature can only be used in thermal runs.

### Examples

**Example 1**

The first item of the **ASPHALTE** keyword is **WEIGHT**, and precipitation set as a function of pressure and temperature.

```
ASPP2P
 -- pressure  temperature
  'P'    'T'  /
```

**Example 2**

The first item of the **ASPHALTE** keyword is **WEIGHT**, and precipitation set as a function of temperature and molar fraction of the 2nd component

```
ASPP2P
 -- temperature concentration 2nd component
  'T'    'Z'      2  /
```
ASPPW2D  Asphalten two parameters precipitation data

This keyword provides data for the asphaltene precipitation in the case where the characterization is based on the percentage molar weight of specified component(s). The characterization is specified by the first item of the ASPHALTE keyword, and the component range is specified by the ASPFLOC keyword.

This keyword takes a 2D table where the first row is either the temperature or the molar fraction and the first column is either the pressure or the temperature depending on the couple of variables specified in the ASPP2P keyword. The other values in the table represent the percentage molar weight of asphaltene component(s) dissolved in the oil phase and should lie between 0 and 100. The table should be terminated by a slash (/).

The number of columns and rows for the table can be set using items 21 and 22 of the TABDIMS keyword respectively. If defaulted the table can take 5 columns and 5 rows.

For more details please refer to the "Asphaltene option" in the ECLIPSE Technical Description.

Example

The first item of the ASPHALTE keyword is WEIGHT (for illustrative purposes):

```
ASPP2P
--- pressure  temperature
    'P'     'T' /
ASPPW2D
------------- temperature -------------
100   150   200   400 /
---pressure
1000           10.0   30.0   50.0   60. /
2000           5.0   20.0   30.0   50. /
3000          10.0   25.0   45.0   80. /
8000          80.0   80.0   90.0   100. /
/```


Asphaltene as percentage weight

This keyword is set to provide data for the asphaltene precipitation in the case where the characterization is based on the percentage molar weight of specified component(s). The characterization is specified by the first item of the ASPHALTE keyword, and the component range is specified by the ASPFLOC keyword.

This keyword should be used in conjunction with ASPP1P keyword to specify the variable the molar weight percentage limit is set against. If the ASPP1P keyword is not specified, the default variable is set to pressure.

This keyword provides the percentage molar weight dissolved in solution as a function of a chosen variable (pressure, temperature or molar fraction). For more details please refer to the "Asphaltene option" in the ECLIPSE Technical Description.

Column:

1. Pressure (oil pressure), or temperature or molar fraction.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
   UNITS: dimensionless
   Values should increase monotonically down the column.

2. The corresponding percentage of “asphaltene” dissolved in the oil phase.
   Values should be between 0 and 100. A value of 1.6 means that the oil phase can contain a maximum of 1.6% asphaltene by weight. The number of rows can be specified in item 18 of TABDIMS.

   No extrapolation is made if the pressure goes outside the range specified in the table. The lowest or highest percentage of asphaltene will be used accordingly.

Examples

Example 1

The first item of the ASPHALTE keyword is WEIGHT, and precipitation set as a function of pressure:

<table>
<thead>
<tr>
<th>ASPP1P</th>
<th>pressure</th>
<th>% weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>p</td>
<td>3000</td>
<td>30.0</td>
</tr>
<tr>
<td></td>
<td>3500</td>
<td>70.0</td>
</tr>
<tr>
<td></td>
<td>4000</td>
<td>100.0</td>
</tr>
</tbody>
</table>

Example 2

The first item of the ASPHALTE keyword is WEIGHT, and precipitation set as a function of the molar fraction of the 2nd component.

<table>
<thead>
<tr>
<th>ASPP1P</th>
<th>molar fraction 2nd component</th>
<th>% weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z2</td>
<td>0.001</td>
<td>100.0</td>
</tr>
<tr>
<td>Value</td>
<td>70.0</td>
<td></td>
</tr>
<tr>
<td>-------</td>
<td>------</td>
<td></td>
</tr>
<tr>
<td>0.010</td>
<td>10.0</td>
<td></td>
</tr>
</tbody>
</table>
Asphaltene oil viscosity change

This keyword is set to provide data for the modeling of the oil viscosity change when asphaltene precipitation occurs (see item 3 in ASPHALTE keyword). There are 3 possible ways to model the oil viscosity change:

1. Generalized Einstein model (one parameter): slope of relative viscosity with respect to concentration.

2. Krieger and Dougherty model (two parameter): mass concentration at maximum packing, and intrinsic viscosity.

3. Look-up table given the oil viscosity multiplier as a function of the mass fraction of the asphaltene precipitate. The first column is the mass fraction and the second column is the viscosity multiplier. The number of rows can be set in item 19 of TABDIMS.

No extrapolation is made if the mass fraction of precipitate goes outside the range specified in the table. The lowest or highest viscosity multiplier will be used accordingly.

Depending on the model chosen, this keyword takes the appropriate number of data, as shown in the examples below.

For more details please refer to the "Asphaltene option" in the ECLIPSE Technical Description.

Examples

Example 1

Generalized Einstein mode: EI in item 3 in the ASPHALTE keyword.

```
ASPVISO
  -- slope
  2.5 /
```

Example 2

Krieger and Dougherty model: KD in item 3 in the ASPHALTE keyword.

```
ASPVISO
  -- Conc. max pack. int. visco
  0.7 2 /
```

Example 3

Look-up table: TAB string in item 3 in the ASPHALTE keyword.

```
ASPVISO
  -- mass.frac multiplier
  0.00 1.0
  0.01 1.01
  0.05 1.06 /
```
Asphaltene wettability factor data

This keyword can be used to define a wettability factor $F$ as a function of the asphaltene deposition. The wettability factor is used to interpolate relative permeability between the water wet relative permeabilities at $F = 0$, provided by standard keywords such as `SWFN`, `SOF3` or `SWOF` and the oil wet relative permeabilities at $F = 1$, provided by the `ASPKROW` keyword.

The keyword is followed by a number of tables, each containing two columns of data:

1. The volume fraction of asphaltene deposit, either $S_s$ or $\varepsilon$.

   If the solid model is used for asphaltene deposition, the volume fraction is measured as a proportion of the pore volume, and the solid saturation $S_s$ is required here. However, if the pre-2011.1 asphaltene deposition model is used, the volume fraction of asphaltene deposit is measured as a proportion of the cell volume and $\varepsilon = \varphi \cdot S_s$ is required here.

2. The wettability factor $F$ which must be between 0 and 1.

The number of tables to be provided with this keyword is equal to the number of saturation tables (see item 1 of the `TABDIMS` keyword), and the maximum number of rows in each table is set by item 23 of the `TABDIMS` keyword.

For more details please refer to the "Asphaltene option" in the ECLIPSE Technical Description.

**Note:** This keyword is active only where the system is water wet, defined here by the oil-water capillary pressure $P_{cow} > 0$

### Example

```
ASPWETF
-- deposit F factor
  0.00  0.0
  0.01  0.01
  0.10  0.20 /
```
AUTOCOAR  Specifies a box of cells for grid coarsening within an auto refinement

This keyword is analogous to the COARSEN keyword and may be used in auto-refined grids (keyword AUTOREF specified in RUNSPEC). The AUTOCOAR keyword has two effects. Firstly it may be used to coarsen cells in the base grid, in a similar manner to COARSEN. Secondly, it prevents auto refinement from happening within the box of cells specified in items 1 - 6; the coarse cells in the base grid are retained.

The difference between the effects of the COARSEN and AUTOCOAR keywords is that any coarsened cell amalgamations constructed with COARSEN is then auto refined into the number of cells specified in AUTOREF, while those constructed with AUTOCOAR remain unrefined.

Note that it is possible to keep the original base cell structure within a box of cells by using AUTOCOAR to coarsen them into the same number of coarse cells. The effect of this is to keep them unaffected by the auto refinement process.

The AUTOCOAR keyword is followed by any number of records, each containing the following items and each terminated with a slash (/). The set of records must end with a blank record containing just a slash. Each record defines a box of grid cells for coarsening. The data items are:

1.   \text{I1} Lower I coordinate of the box.
2.   \text{I2} Upper I coordinate of the box.
3.   \text{J1} Lower J coordinate of the box.
4.   \text{J2} Upper J coordinate of the box.
5.   \text{K1} Lower K coordinate of the box.
6.   \text{K2} Upper K coordinate of the box.
7.   \text{NX} Number of coarsened cells along the X direction in this box.
   \text{NX} must be an exact factor of \text{I2-I1+1}.
8.   \text{NY} Number of coarsened cells along the Y direction in this box.
   \text{NY} must be an exact factor of \text{J2-J1+1}.
9.   \text{NZ} Number of coarsened cells along the Z direction in this box.
   \text{NZ} must be an exact factor of \text{K2-K1+1}.

This divides the box \text{I1-I2, J1-J2, K1-K2} into \text{NX * NY * NZ} coarse cells. Coarsening boxes specified using COARSEN or AUTOCOAR must not overlap one another, but they may join together.

The parameter \text{MCOARS} in keyword LGR in the RUNSPEC section must be set equal to or greater than the total number of coarse cells. Coarsening generates non-neighbor connections, and is disallowed if the keyword NONNC has been entered in RUNSPEC.

Examples

Example 1

Keyword syntax
Example 2

Consider a 6x6x1 grid with:

```
AUTOCAR
  -- I1-I2  J1-J2  K1-K2  NX NY NZ
   1 10  1  4  1  6  5  2  3 /
   1  4  5 10  1  6  2  3  3 /
```

```
Example 2
Consider a 6x6x1 grid with:

COARSEN
  -- I1 I2 J1 J2 K1 K2 NX NY NZ
   1  6  1  2  1  1  3  1  1 /
   1  2  3  4  1  1  1  1  1 /
   5  6  3  4  1  1  1  1  1 /
   1  6  5  6  1  1  3  1  1 /
```

With the original uncoarsened cells shown by dashed lines, this gives a grid like figure 3.1.

```
Figure 3.1. The effect of the COARSEN keyword
```

Now with

```
AUTOREF
  3 3 1 /
```

the grid is refined as shown in figure 3.2.
Figure 3.2. The effect of the AUTOREF keyword

But if the COARSEN keyword is replaced by AUTOCOAR, such as:

```
AUTOCOAR
  - I1 I2 J1 J2 K1 K2 NX NY NY
  1  6  1  2  1  1  3  1  1  /
  1  2  3  4  1  1  1  1  1  /
  5  6  3  4  1  1  1  1  1  /
  1  6  5  6  1  1  3  1  1  /
```

then the coarsened cells are not auto refined and the resultant grid appears as shown in figure 3.3.
Figure 3.3. The effect of the AUTOCOAR keyword
Set options for auto refinement

This keyword sets options for auto refinement.

The data consists of the following items specifying the refinement factors to be applied to the grid in each direction. The data must be terminated by a slash (/).

1. Refinement factor in the X-direction; NX must be an odd number.
   DEFAULT: 1
2. Refinement factor in the Y-direction; NY must be an odd number.
   DEFAULT: 1
3. Refinement factor in the Z-direction; NZ must be an odd number.
   DEFAULT: 1
4. Option for applying transmissibility multipliers.
   0 Any transmissibility multipliers for example MULTX are inherited from host cells in the base grid.
   1 Transmissibility multipliers, for example MULTX, on the base grid are only applied on the auto-refined grid at boundaries of the base grid.
   2 Allows additional transmissibility multipliers SMULTX, SMULTY and SMULTZ which are applied only at boundaries of the base grid as in option 1. Any multipliers MULTX, MULTY or MULTZ are inherited from the host cells in the base grid as in option 0 above.
   DEFAULT: 0

The auto refinement facility is useful for automatic scaling of datasets for massively parallel computations, and for grid refinement sensitivity studies. (See “Automatic refinement (ECLIPSE 100)” of the ECLIPSE Technical Description for more details.)

If auto refinement is used, then neither PETGRID nor GDFILE can be specified.

Example

Refine each cell into 3 x 3 areally:

```
AUTOREF
   --NRX  NRY  NRZ  Option for MULTX/Y/Z
   3    3    1    1             /
```

Keywords

394
AUTOSAVE  Specify automatic writing of SAVE files

This keyword specifies that SAVE files are to be written every N steps during the simulation. It may be entered more than once in the SCHEDULE section.

The effect depends on the MULTSAVE option specified in the RUNSPEC section:

- If MULTSAVE set to -1: No effect, as SAVE files suppressed
- If MULTSAVE set to 0 (default): Continuously overwrites SAVE files
- If MULTSAVE set to 1: Produce a new SAVE file each time.

Example

```
AUTOSAVE
1 /
```
This section contains an alphabetic listing of keywords starting with the letter B.
BAKER1 Request Baker three-phase relative permeability model (General)

This keyword, which is followed by a line containing up to three items terminated with a slash (/), is used to specify that the three-phase relative permeability values are to be calculated using Baker’s general formulation. See "Saturation functions" in the ECLIPSE Technical Description for a description of three-phase relative permeability models available in ECLIPSE. The BAKER1 keyword should only be used in three-phase runs.

The possible item values are

- **OIL**: Baker oil relative permeability. Baker oil relative permeability with two-phase relative permeability hysteresis specified in EHYSTR.
- **GAS**: Baker gas relative permeability. Baker gas relative permeability with two-phase relative permeability hysteresis specified in EHYSTR.
- **WATER**: Baker water relative permeability. Baker water relative permeability with two-phase relative permeability hysteresis specified in EHYSTR.

The options are used to determine whether or not to use Baker’s calculation for each of the water, gas and oil phases, respectively providing independent control on a per phase basis. The items can appear in any order and are optional. At least one of the above values should be specified (see Examples 1, 2, and Example 3).

The BAKER1, BAKER2, IKU3P and ODD3P relative permeability models are mutually exclusive. The BAKER1, BAKER2, STONE, STONE1 and STONE2 relative permeability models are mutually exclusive for the oil phase. The use of BAKER1 requires the second family of saturation function keywords, that is, SWFN or SWF3, SGFN or SGF3 and SOF3. If the Baker water model is specified the SWF3 keyword must be used. If the Baker model is specified, but not for water, the SWFN keyword must be used. Similarly, if the Baker gas model is specified the SGF3 keyword must be used. If the Baker model is specified, but not for gas, the SGFN keyword must be used.

**Note:** The hysteresis model selected according to the items on the EHYSTR keyword for the three-phase relative permeability will be used for omitted phases. See "Saturation functions" in the ECLIPSE Technical Description for a description of default three-phase relative permeability models available in ECLIPSE.

### Examples

**Example 1**

Example 1 implements the BAKER1 model for water and gas three-phase relative permeability.

```plaintext
BAKER1 WATER GAS/
```

**Example 2**

Example 2 shows how to use the BAKER1 model for gas and oil three-phase relative permeability.
Example 3

Example 3 uses the BAKER1 model for water, gas and oil three-phase relative permeability. Water, gas and oil hysteresis is deselected if hysteresis model was turned on.

BAKER1
WATER GAS OIL/
BAKER2 Request Baker three-phase relative permeability model (Simplified)

This keyword, which is followed by a line containing up to three items terminated with a slash (/), is used to specify that the three-phase relative permeability values are to be calculated using Baker’s simplified formulation. See “Saturation functions” in the ECLIPSE Technical Description for a description of three-phase relative permeability models available in ECLIPSE. The BAKER2 keyword should only be used in three-phase runs.

The possible item values are

- **OIL**  Baker oil relative permeability. Baker oil relative permeability with two-phase relative permeability hysteresis specified in EHYSTR.

- **GAS**  Baker gas relative permeability. Baker gas relative permeability with two-phase relative permeability hysteresis specified in EHYSTR.

- **WATER** Baker water relative permeability. Baker water relative permeability with two-phase relative permeability hysteresis specified in EHYSTR.

The options are used to determine whether or not to use Baker’s calculation for each of the water, gas and oil phases, respectively providing independent control on a per phase basis. The items can appear in any order and are optional. At least one of the above values should be specified (see Examples 1, 2, and Example 3).

The BAKER1, BAKER2, IKU3P and ODD3P relative permeability models are mutually exclusive. The BAKER1, BAKER2, STONE, STONE1 and STONE2 relative permeability models are mutually exclusive for the oil phase. The use of BAKER2 requires the second family of saturation function keywords, that is, SWFN or SWF3, SGFN or SGF3 and SOF3. If the Baker water model is specified the SWF3 keyword must be used. If the Baker model is specified, but not for water, the SWFN keyword must be used. Similarly, if the Baker gas model is specified the SGF3 keyword must be used. If the Baker model is specified, but not for gas, the SGFN keyword must be used.

**Note:** The hysteresis model selected according to the items on the EHYSTR keyword for the three-phase relative permeability will be used for omitted phases. See “Saturation functions” in the ECLIPSE Technical Description for a description of default three-phase relative permeability models available in ECLIPSE.

**Examples**

**Example 1**

Example 1 describes BAKER2 model for water and gas three-phase relative permeability.

```
BAKER2 WATER GAS/
```

**Example 2**

Example 2 describes BAKER2 model for water, gas and oil three-phase relative permeability.
Example 3 describes the \texttt{BAKER1} model for water and oil three-phase relative permeability, and \texttt{BAKER2} for gas three-phase relative permeability.
**BDENSITY**

**Brine surface density**

This keyword may optionally be used in runs with the brine tracking option active (keyword BRINE). It enables the brine surface density to vary with the salt concentration. If the keyword is not present, the brine takes the surface density of water, as specified in the DENSITY (or GRAVITY) keyword.

The data comprises NTPVT (see keyword TABDIMS) records, each terminated by a slash (/). Each record consists of up to NPPVT values of water surface density. The density values in each record correspond with the salt concentration values tabulated in column 1 of each table in the PVTWSALT keyword.

**UNITS:** kg/m$^3$ (METRIC), lb/ft$^3$ (FIELD), gm/cc (LAB), kg/m$^3$ (PVT-M)

See also keywords PVTWSALT and DENSITY.

**Example**

With NTPVT = 2 and NPPVT ≥ 4 in TABDIMS:

```plaintext
BDENSITY
  64.0  65.0  67.0  69.0  /
  64.1  64.8  65.1 /
PVTWSALT
  5000  0.0 /
  0.0  1.01  3.00E-06  0.52  0.0
  0.2  1.00  3.00E-06  0.52  0.0
  0.5  0.98  3.00E-06  0.52  0.0
  1.0  0.95  3.00E-06  0.52  0.0 /
  4000  0.0 /
  0.0  1.01  3.00E-06  0.52  0.0
  0.2  1.00  3.00E-06  0.52  0.0
  0.5  0.98  3.00E-06  0.52  0.0 /
```

**Keywords**

ECLIPSE Reference Manual
Variation of saturated gas FVF with pressure and Gi

The data comprises NTPVT (see keyword TABDIMS) tables of saturated gas FVF multiplier functions, each terminated by a slash (/), describing the variation of the saturated FVF of gas with pressure and Gi values. Each table consists of up to NPPVT sets of data records. Each set of records within a table consists of up to NPPVT records of data items to allow the specification of a two-dimensional (P, Gi) function which multiplies the saturated gas FVF value at (P, Gi=0) to give the true saturated gas FVF at (P, Gi). The saturated gas FVF value at (P, Gi=0) is obtained from the corresponding PVTG table.

The first record of each elementary set of records contains the following items of data:

1. The gas pressure value.
   The values should increase monotonically down the column.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB)

2. The factor by which the saturated gas FVF calculated from the PVTG keyword at a pressure corresponding to item 1 has to be multiplied to give the true saturated gas FVF at a pressure corresponding to item 1 and a Gi value given by item 1 in the corresponding GINODE table.

For record 2 and all subsequent records up to a maximum of NPPVT, there should be only 1 data item in the record. This item defines the corresponding factor by which the saturated gas FVF at (P, Gi=0) (computed from the PVTG data) has to be multiplied to give the true saturated FVF for gas at a pressure value given in item 1 above and whose Gi value is specified in the corresponding record in the GINODE keyword.

See "Gi pseudo-compositional model" in the ECLIPSE Technical Description for further information.

Example

With NTPVT =1 and NPPVT =2

<table>
<thead>
<tr>
<th>P</th>
<th>Gi</th>
<th>Multiplier</th>
</tr>
</thead>
<tbody>
<tr>
<td>100.0</td>
<td>1.0</td>
<td>0.99</td>
</tr>
<tr>
<td>200.0</td>
<td>1.0</td>
<td>0.992</td>
</tr>
</tbody>
</table>

Keywords

402
Binary interaction coefficients

In a run with \( N_c \) components, using an equation of state, this keyword defines the binary interaction coefficients between the components.

The keyword should be followed by \( N_c(N_c-1)/2 \) values, where \( N_c \) is the number of components specified in the RUNSPEC section with the COMPS keyword. The values are arranged in lower triangular form, the symmetry of the interaction coefficients being used to define the upper triangular part, and the self binary interaction coefficients being zero.

The use of the binary interaction coefficients in the equation of state is discussed in "Equations of state" in the ECLIPSE Technical Description.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. This is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

DEFAULT: 0 (for all values)

Examples

Example 1

For a five-component system with a binary interaction coefficient of 0.01 between component pairs 1-2 and 1-3; 0.02 between components 1-4; 0.005 between pairs 2-3 and 4-5; and 0.001 between pairs 1-5, 3-4 and 3-5.

```
  BIC
  0.01
  0.01  0.005
  0.02  0.0  0.001
  0.001 0.0  0.001 0.0005
```

Example 2

With two reservoir EoS regions:

```
  BIC
  -0.0200
  0.1000  .0360
  -0.022
  0.11    .0560
```
In GASWAT mode the binary interaction coefficients for the Peng-Robinson equation of state are different from those used for the hydrocarbon phases, and may be temperature dependent. The default forms used are those suggested by Soreide and Whitson, and are described in "GASWAT option" in the ECLIPSE Technical Description. The general form is:

\[ k_{jw}^a = bq_1 + bq_2 T_{rj} + bq_3 T_{rj}^2 \quad \text{Eq. 3.11} \]

The BICAQ1 keyword allows \( bq_1 \) values to be entered directly. The form is the same as that for the BIC keyword.

\( bq_2 \) and \( bq_3 \) default values may be overridden using the related BICAQ2 and BICAQ3 keywords.
BICAQ2 Second binary interaction coefficient for aqueous phase

This keyword allow you to enter a non-default aqueous phase binary interaction coefficient for use with the GASWAT option. See keyword BICAQ1 for further details.
BICAQ3  Third binary interaction coefficient for aqueous phase

This keyword enters a non-default aqueous phase binary interaction coefficient for use with the GASWAT option. See keyword BICAQ1 for further details.
Binary interaction coefficients for surface

In a run with N_c components, using an equation of state, this keyword defines the binary interaction coefficients between the components. These are the coefficients to be used for the production system (that is on the surface). If the BICS keyword is not used, the values supplied with keyword BIC is used.

The format of the keyword is exactly the same as that of BIC.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.
BIGMODEL  Allows large models to run

Note: If this keyword is included in a dataset, it can safely be removed.

This keyword is ignored as ECLIPSE uses the memory allocator of the host operating system.
BIOTC

**BIOTC**

**Biot’s constant for rock/fluid interaction**

This keyword has no effect if **GEOMECH** is not specified in the **RUNSPEC** section.

The keyword should be followed by one non-negative real number for every grid block in the current input box, specifying the Biot constant. This parameter is dimensionless and must be greater than the porosity and less than or equal to 1. The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0.4). Note that spaces may not be inserted on either side of the asterisk.

The default is 1.0 for every block.

**Example**

```
BIOTC
324*0.3 /
```
**BIOTCTAB**  
**Geomechanics Biot constant multiplier as a tabular function of rock stress**

A multiplier for Biot’s constant may be specified in table form as a function of maximum and minimum principal stress. This keyword is only valid if the geomechanics option is chosen (see `GEOMECH` in the `RUNSPEC` section).

The data comprises `NGMYRG` (first item of keyword `GEODIMS` in the `RUNSPEC` section) tables, each terminated by a slash (`/`), where `NGMYRG` is the number of geomechanics regions in the model.

The first record sets the values of the maximum principal stress, $\sigma_1$. These must be monotonically increasing. This record must end with a slash (`/`).

**Record 1**

`$\sigma_{1,1} \quad \sigma_{1,2} \cdots \quad \sigma_{1,NGMBTC}$`

The next `NGMBTR` records set the values of the minimum principal stress, $\sigma_3$, followed by the table data for the Biot constant multipliers $BM$. A value of 1 for a multiplier means the Biot constant is unchanged. Values of the minimum principal stress must be monotonically increasing; that is subsequent records must have increasing $\sigma_3$. Repeat counts may be used with the multipliers, but not with the maximum/minimum principal stresses.

**Record 2**

`$\sigma_{3,1} \quad BM_{1,1} \cdots \quad BM_{1,NGMBTC}$`

**Record 3**

`$\sigma_{3,2} \quad BM_{2,1} \cdots \quad BM_{2,NGMBTC}$`

... ... ...

... ... ...

**NGMBTR+1**

`$\sigma_{3,NGMBTR} \quad BM_{NGMBTR,1} \cdots \quad BM_{NGMBTR,NGMBTC}$`

Records 2 through `NGMBTR` must end with a slash (`/`).

**Note:** If the Biot constant multiplier is only a function of the maximum principal stress, then set or default the number of rows, `NGMBTR`, to be 1. In this case, the first value in Record 2, $\sigma_{3,1}$, is ignored. Similarly, if the multiplier is only a function of the minimum principal stress, then set or default the number of columns, `NGMBTC`, to be 1. Again, in this case the first value in Record 1, $\sigma_{1,1}$, is ignored.

The set of records is ended by a blank record, containing only a slash.

The Biot constant multipliers are not checked for monotonicity. If an input maximum and minimum principal stress from the simulation model lies outside the range of the above table, constant slope extrapolation is used.

See "Geomechanics" in the *ECLIPSE Technical Description*.

**Example**

With `NGMYRG=1, NGMBTC=4, NGMBTR=3`:

---

**Keywords**

<table>
<thead>
<tr>
<th>ECLIPSE 100</th>
<th>ECLIPSE 300</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>SPECIAL</td>
</tr>
<tr>
<td>RUNSPEC</td>
<td>GRID</td>
</tr>
<tr>
<td>EDIT</td>
<td>PROPS</td>
</tr>
<tr>
<td>REGIONS</td>
<td>SOLUTION</td>
</tr>
<tr>
<td>SUMMARY</td>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>

---

410
BIOTCTAB

-- Maximum principal stresses
  5000   7000   9000   11000  /
-- Minimum principal stress, Biot constant multipliers
  5000   2*1   0.99   0.98  /
  6000   1     0.99   0.97   0.94  /
  8000   0.99   0.95   0.93   0.91  /
/

Keywords
BLACKOIL Requests black oil mode

This keyword activates the black oil mode, and is an alternative to COMPS. Once BLACKOIL has been specified, keywords appropriate to compositional mode may not be entered. If neither COMPS nor BLACKOIL are specified, black oil mode will be assumed.

In ECLIPSE 300, if BLACKOIL is specified, the presence of all three phases (oil, water and gas) is assumed. A reduced phase black oil model (for example, oil and water only) can be selected by omitting the BLACKOIL keyword and specifying the phases which are present via the phase keywords (WATER, OIL and GAS). However, it is strongly recommended that if a reduced phase black oil model is required, this is simulated instead by specifying BLACKOIL and selecting properties for the unrequired third phase such that it is not active in the simulation.

In ECLIPSE 100 the BLACKOIL keyword tries to reproduce the ECLIPSE 300 default BLACKOIL behavior by setting the following keywords: GAS, DISGAS, VAPOIL (if PVTG is entered).

In ECLIPSE 300, the default solution method for black oil simulations is FULLIMP.

This keyword can be used with the Thermal option as an alternative to LIVEOIL and DEADOIL. The simulation will be run using the Thermal code, but properties should be specified using black oil keywords.
BOGI Variation of saturated oil FVF with pressure and Gi

The data comprises NTPVT (see keyword TABDIMS) tables of saturated oil FVF multiplier functions, each terminated by a slash(/), describing the variation of saturated FVF of oil with pressure and Gi values. Each table consists of up to NPPVT sets of data records. Each set of records within a table consists of up to NPPVT records of data items to allow the specification of a two-dimensional (P, Gi) function which multiplies the saturated oil FVF value at (P, Gi=0) to give the true saturated oil FVF at (P, Gi). The saturated oil FVF value at (P, Gi=0) is obtained from the corresponding PVTO table.

The first record of each elementary set of records contains the following items of data:

1. The oil pressure value.
   The values should increase monotonically down the column.
   UNITS: barsa (METRIC), psia (FIELD), atm (LAB)

2. The factor by which the saturated oil FVF calculated from the PVTO keyword at a pressure corresponding to item 1 has to be multiplied to give the true saturated oil FVF at a pressure corresponding to item 1 and a Gi value given by item 1 in the corresponding GINO keyword.

For record 2 and all subsequent records up to a maximum of NPPVT, there should be only 1 data item in the record. This item defines the corresponding factor by which the saturated oil FVF at (P, Gi=0) (computed from the PVTO data) has to be multiplied to give the true saturated FVF for oil at a pressure value given in item 1 above and whose Gi value is specified in the corresponding record in the GINO keyword.

See "Gi pseudo-compositional model" in the ECLIPSE Technical Description for further information.

Example

With NTPVT =1 and NPPVT =2

| BOGI | 100.0 | 1.0  | 0.97 | / |
|      | 200.0 | 1.0  | 0.952| / |

Keywords

413
BOUNDARY  Defines area of grid to be printed

The BOUNDARY keyword may be used to specify the rectangular area of the grid for which printed output of grid data or solution data (for example TOPS, PERMX, PORV, FIPNUM and SWAT) is required.

The keyword should be followed by the data items described below. The initial values are also shown:

1. First cell on X axis to be included in output (IX1)
   INITIAL: 1
2. Last cell on X axis to be included in output (IX2)
   INITIAL: NDIVIX (which specified in RUNSPEC keyword DIMENS)
3. First cell on Y axis to be included in output (JY1)
   INITIAL: 1
4. Last cell on Y axis to be included in output (JY2)
   INITIAL: NDIVIY (which specified in RUNSPEC keyword DIMENS)
5. First cell on Z axis to be included in output (KZ1)
   INITIAL: 1
6. Last cell on Z axis to be included in output (KZ2)
   INITIAL: NDIVIZ (which specified in RUNSPEC keyword DIMENS)
7. Orientation Index
   1  X across the page, Y down the page, for each Z (XYZ)
   2  X across the page, Z down the page, for each Y (XZY)
   3  Y across the page, X down the page, for each Z (YXZ)
   4  Y across the page, Z down the page, for each X (YZX)
   5  Z across the page, X down the page, for each Y (ZXY)
   6  Z across the page, Y down the page, for each X (ZYX)

If the orientation index is set negative (for example -1, -2 or -3), and the number of cells in the inner direction exceeds 15, then all the data for the inner directions are printed prior to moving to the next outer layer.

For example, setting the orientation index to -1 prints all the XY data for layer 1, followed by all the XY data for layer 2, and so on sequentially for the other layers.

DEFAULT: 1

8. Dual Porosity output flag (relevant in dual porosity runs only)
   BOTH  Both matrix and fracture cells
   MATRIX  Matrix cells only
FRACTURE  Fracture cells only

The word may be abbreviated, as only the first character is significant.

DEFAULT: BOTH

The data must satisfy:

\[ 1 \leq IX1 \leq IX2 \leq NDIVIX \]
\[ 1 \leq JY1 \leq JY2 \leq NDIVIY \]
\[ 1 \leq KZ1 \leq KZ2 \leq NDIVIZ. \]

The data field should be terminated by a slash (/).

If fewer than eight items are read, the remainder are left unchanged.

**ECLIPSE 100**

In addition, this keyword controls the orientation of the output, and in dual porosity run enables you to print only the matrix or fracture cells.

**ECLIPSE 300**

This keyword can also be used in the RUNSPEC, PROPS and SUMMARY sections. Output to GRAF specified by OUTSOL and in the SUMMARY section is not affected.

**Example**

```
-------- IX1-IX2  JY1-JY2  KZ1-KZ2 Orientation Dual Poro
BOUNDARY  3  7  1  8  3  3 /
```
Redefines the current input box

The keyword should be followed by six integers that redefine the current input box. Subsequent operations using \texttt{EQUALS}, \texttt{ADD}, \texttt{MULTIPLY} and \texttt{COPY} only alter grid blocks within the current input box. Similarly, data read into an array (for example, using \texttt{PERMX}, \texttt{PORV} or \texttt{SATNUM}) are assigned to the grid blocks in the current input box.

The keyword should be followed by the data items described below, terminated with a slash.

1. First block on X axis of the new input box (\texttt{IX1})
2. Last block on X axis of the new input box (\texttt{IX2})
3. First block on Y axis of the new input box (\texttt{JY1})
4. Last block on Y axis of the new input box (\texttt{JY2})
5. First block on Z axis of the new input box (\texttt{KZ1})
6. Last block on Z axis of the new input box (\texttt{KZ2})

The data must satisfy:

\begin{align*}
1 & \leq \texttt{IX1} \leq \texttt{IX2} \leq \texttt{NDIVIX} \\
1 & \leq \texttt{JY1} \leq \texttt{JY2} \leq \texttt{NDIVIY} \\
1 & \leq \texttt{KZ1} \leq \texttt{KZ2} \leq \texttt{NDIVIZ} \quad \text{(where \texttt{NDIVIX}, \texttt{NDIVIY} and \texttt{NDIVIZ} are specified in RUNSPEC keyword \texttt{DIMENS})}
\end{align*}

The default cell limits are the current \texttt{BOX} values.

The data should be terminated by a slash (\texttt{/}).

Note that at the end of each input section, the program effectively supplies an \texttt{ENDBOX} and resets the input box.

This keyword can be used in the \texttt{RUNSPEC} and \texttt{SUMMARY} sections.

See also keywords \texttt{ADD}, \texttt{COPY}, \texttt{ENDBOX}, \texttt{EQUALS}, \texttt{MULTIPLY}, \texttt{MINVALUE} and \texttt{MAXVALUE}.

Example

\begin{verbatim}
BOX
--- IX1-IX2  JY1-JY2  KZ1-KZ2
   3    7   13  3  5 /
\end{verbatim}
This keyword enables the Block Parallel license option.

This is an alternative license option for parallel simulations with up to eight parallel domains. In such cases, a single Block Parallel license can be used instead of the (up to) eight parallel licenses that would normally be required.

To use this option the parallel keyword (PARALLEL) must also be present, with item 1 (the number of domains) set to a value in the range 2 to 8.

The keyword BPARA contains no items of associated data.

Note:

- The Block Parallel license feature is not compatible with multiple realization (MULTREAL) runs.
- Only one Block Parallel license can be used in an ECLIPSE run. If more than eight domains are requested the simulation will stop with an error message and the simulation should then be run using the normal parallel licensing options.
- The BPARA keyword cannot be used with the Reservoir Coupling option in ECLIPSE 100.

See "Block parallel license" in the ECLIPSE Technical Description for more information.

Example

```plaintext
BPARA
PARALLEL
--NDMAIN MACHINE TYPE
  8 DISTRIBUTED /
```
Dimensions for interpolated block quantities

The data consists of items which specify the number of SUMMARY keywords for interpolated quantities. Such SUMMARY keywords include BPR_X, BHD_X, BHDF_X, BSCN_X and BCTRA_X for the global grid, and LBPR_X, LBHD_X, LBHDF_X, LBSCN_X and LBCTRA_X for local grids. The data must be terminated by a slash (/).

1. **MXNBIP**
   - The maximum number of lines of block interpolated data entered in the SUMMARY section for the global grid, using keywords BPR_X, BHD_X, BHDF_X, BSCN_X and BCTRA_X.
   - DEFAULT: 10

2. **MXNLBI**
   - The maximum number of lines of block interpolated data entered in the SUMMARY section for all local grids, using keywords LBPR_X, LBHD_X, LBHDF_X, LBSCN_X and LBCTRA_X.
   - DEFAULT: 1

**Example**

A maximum of twenty listings for the global grid, and ten listings for local grids.

```
BPIDIMS
20 10 /
```
BRANPROP  Defines the extended network branches

The keyword must be used to define the structure of the network when the extended network model is employed (keyword NETWORK in the RUNSPEC section). The network structure can be different from the group control hierarchy defined by keyword GRUPTREE. However, it must have a gathering tree structure, and the source nodes (the bottom nodes in the tree) must correspond to groups in the GRUPTREE hierarchy (usually well groups or satellite groups); in ECLIPSE 100 the corresponding nodes and groups must have the same name in both structures. The remaining network nodes and GRUPTREE groups need not coincide.

You cannot mix BRANPROP and GRUPNET; the complete production network must be defined with either BRANPROP and NODEPROP (for an extended network) or GRUPNET (for a standard network).

In ECLIPSE 300, if an injection network is present as well, it must be of the same type as the production network. If the production network has been defined with BRANPROP and NODEPROP then the injection network must also be defined with these keywords.

In ECLIPSE 100 only production networks may use BRANPROP and NODEPROP; injection networks must be defined using GNETINJE.

The BRANPROP keyword is followed by any number of records, each containing the following items of data and terminated with a slash (/). The set of records must end with a blank record, containing only a slash.

1. Name of the branch’s downtree node.
   (up to 8 characters)
   This is the node that is nearer to the wells and further from the terminal node at the top of the tree. In a production network it is the branch’s inlet node; in an injection network it is the branch’s outlet node.

2. Name of the branch’s uptree node.
   (up to 8 characters)
   This is the node that is nearer to the terminal node at the top of the tree. In a production network it is the branch’s outlet node; in an injection network it is the branch’s inlet node.

3. VFP table number for the branch.
   In a production network this should be the production VFP table number (keyword VFPPROD).
   In an injection network this should be the injection VFP table number (keyword VFPINJ). The inlet and outlet nodes must both belong to the same type of network (Item 7 of keyword NODEPROP).

   Two or more network pipelines may share the same VFP table if they have similar pressure loss characteristics.

   A value of 9999 can be entered to indicate that there is no pressure loss in the network branch defined by items 1 and 2. Branches representing chokes (see item 3 of keyword NODEPROP) must have a value of 9999 input here.

4. Artificial Lift Quantity (ALQ) used in the pressure loss calculations for the branch.

Compositional production VFP tables (keyword VFPPROD) are not currently supported in the network model.

A value of 0 will close the branch, that is remove it from the network. The inlet node’s production (or the outlet node’s injection) is not added to the network flows.

Keywords

419
This quantity is used as a look-up parameter for the VFP table in item 3. It may be regarded as the
pump or compressor power, for example, according to the definition used in the construction of the

table.

**ECLIPSE 100**

If the branch contains an automatic compressor (keyword **NETCOMPA**), the ALQ value entered here
should correspond to the compressor’s “off” state (which would normally be zero).

**ECLIPSE 100**

Default this item if the ALQ is equated to **DENO** or **DENG** in item 5.

**UNITS:** These depend on what the ALQ was meant to represent when the table was created. Refer to
section "Record 6:ALQ values" in the description of keyword **VFPPROD**.

**DEFAULT:** Zero

**ECLIPSE 100 only**

5. Equate the branch’s ALQ to the surface density.

The branch’s ALQ may optionally be equated to the average surface density of either the oil phase or
the gas phase flowing along it. **ECLIPSE** will then automatically set it equal to this value when it
looks up the VFP table. The VFP table must have been prepared using the same definition of the ALQ
(see keyword **VFPPROD**), and should span the expected range of surface density values. This option is
useful when a mixture of oil or gas with different surface densities flows into the network. Obviously
the ALQ cannot then also be used to represent a pump or compressor, and the branch cannot contain
an automatic compressor.

**DENO** Set the ALQ equal to the average surface density of the **oil** flowing along the branch.

**DENG** Set the ALQ equal to the average surface density of the **gas** flowing along the branch.

**NONE** Do not equate the ALQ to the oil or gas density. Leave it at the value entered in item 4 (or let
it be reset by the automatic compressor facility).

**DEFAULT:** NONE

End the data record with a slash (/).

End the set of records with a blank record containing just a slash.

See also keyword **NODEPROP**.

**Note:** It is possible to change the network topology during the run by adding new branches between the
nodes. However, each node may only have one uptree branch (in the direction of the terminal node at the
top of the tree). Hence if a node’s uptree branch is redirected to a different uptree node, its original uptree
branch must be removed by giving it a zero VFP table number in item 3.

### Example

<table>
<thead>
<tr>
<th>BRANPROP</th>
<th>UpTre</th>
<th>VFP</th>
<th>ALQ</th>
<th>ALQ=</th>
</tr>
</thead>
<tbody>
<tr>
<td>node</td>
<td>node</td>
<td>table</td>
<td>value</td>
<td>dens?</td>
</tr>
<tr>
<td>GB2</td>
<td>N4</td>
<td>3</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>GA2</td>
<td>N3</td>
<td>2</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>GB1</td>
<td>N2</td>
<td>1</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>GA1</td>
<td>N1</td>
<td>9999</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>N4</td>
<td>N3</td>
<td>4</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>N3</td>
<td>N2</td>
<td>4</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>N2</td>
<td>N1</td>
<td>5</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>N1</td>
<td>FIELD</td>
<td>6</td>
<td>100</td>
<td>/</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Keywords**

420
**Brine tracking is required**

This indicates that the brine tracking option is required, to allow the modeling of waters with different salinities. (See "Brine tracking" in the *ECLIPSE Technical Description*).

For runs using the ECLIPSE polymer flood option (enabled with keyword `POLYMER`), `BRINE` switches on the polymer salt-sensitivity option instead.

The keyword has no associated data for the standard brine tracking option.

When used in conjunction with keyword `ECLMC` the `BRINE` keyword activates the multi-component brine model (ECLIPSE 100 only). In this case the keyword should be followed by a list of salts specified by their chemical formulae. At least one salt must be specified. The salts available are listed in Table 3.2. The data must be terminated by a slash (/)

The Multi-component Brine Model is currently not compatible with the polymer salt-sensitivity option, and will invalidate its keywords.

<table>
<thead>
<tr>
<th>Salt name</th>
<th>Chemical formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sodium Chloride</td>
<td>NaCl</td>
</tr>
<tr>
<td>Potassium Chloride</td>
<td>KCl</td>
</tr>
<tr>
<td>Calcium Chloride</td>
<td>CaCl₂</td>
</tr>
<tr>
<td>Magnesium Chloride</td>
<td>MgCl₂</td>
</tr>
<tr>
<td>Sodium Carbonate</td>
<td>Na₂CO₃</td>
</tr>
<tr>
<td>Potassium Carbonate</td>
<td>K₂CO₃</td>
</tr>
<tr>
<td>Calcium Carbonate</td>
<td>CaCO₃</td>
</tr>
<tr>
<td>Magnesium Carbonate</td>
<td>MgCO₃</td>
</tr>
<tr>
<td>Sodium Sulfate</td>
<td>Na₂SO₄</td>
</tr>
<tr>
<td>Potassium Sulfate</td>
<td>K₂SO₄</td>
</tr>
<tr>
<td>Calcium Sulfate</td>
<td>CaSO₄</td>
</tr>
<tr>
<td>Magnesium Sulfate</td>
<td>MgSO₄</td>
</tr>
</tbody>
</table>

*Table 3.2: List of available salts in the Multi-Component Brine model*

**Example**

*ECLIPSE 100 only*

Multi-component brine model with three salts.

```plaintext
BRINE:
  NACL  CACL2  MGCO3 / `
BTOBALFA  Dual porosity block to block matrix-fracture coupling multiplier

If the Dual Porosity option is being used (keyword DUALPORO in the RUNSPEC section), and the matrix-fracture coupling transmissibilities have been specified in the grid section through the SIGMA or SIGMAV keywords, then this keyword activates additional matrix fracture connections between every active fracture grid block and an active matrix grid block immediately below it. The value specified with this keyword is the multiplier for all these new vertical block-to-block connections equivalent to the value specified by the SIGMA keyword for the conventional dual porosity connections, and represents the fractional contact between the upper fracture and the lower matrix. However, because it is multiplying a directional transmissibility, unlike SIGMA, it is dimensionless.

For further information, see the section on "Block to block transmissibility calculations" in the ECLIPSE Technical Description.

Note that block-to-block connections cannot be made if a multi porosity gridding is specified (using NMATRIX for multi porosity or TRPLPORO for triple porosity).

The keyword should be followed by a single positive real number which will be applied to every grid block in the current grid.

To specify different values for different cells, use the BTOBALFV keyword.

UNITS: dimensionless

Example

```
BTOBALFA
0.01  /
```
BTOBALFV  Dual porosity block to block matrix-fracture coupling multiplier

If the dual porosity option is being used (keyword DUALPORO in the RUNSPEC section), and the matrix-fracture coupling transmissibilities have been specified in the grid section through the SIGMA or SIGMAV keywords, then this keyword activates additional matrix fracture connections between every active fracture grid block and an active matrix grid block immediately below it. The values specified with this keyword are the multipliers for these new vertical block-to-block connections equivalent to the values specified by the SIGMAV keyword for the conventional dual porosity connections, and represent the fractional contact between the upper fracture and the lower matrix. However, because it is multiplying a directional transmissibility, it is dimensionless.

For further information, see the section on "Block to block transmissibility calculations" in the ECLIPSE Technical Description.

Note that block-to-block connections cannot be made if a multi porosity gridding is specified (using NMATRIX for multi porosity or TRPLPORO for triple porosity).

The keyword should be followed by one positive real number for every grid block in the current box within the top NDIVIZ/2 layers of the grid (that is, the matrix cells). Any values input in the bottom half of the grid (the lower NDIVIZ/2 layers) will be ignored.

To specify a single value for all cells, use the BTOBALFA keyword.

UNITS: dimensionless

Example

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX     1 8 1 1 1 2 /
BTOBALFV 0.5 0.5 0.5 0.5 0.5 0.5 /
```
This section contains an alphabetic listing of keywords starting with the letter C.
**CALTRAC**  Sets a tracer to be associated with the calorific value of the gas

This keyword is used to specify the tracer whose concentration represents the calorific value of the gas, when the Gas Calorific Value Control option is being used (see "Gas calorific value control" in the *ECLIPSE Technical Description*).

The tracer must be already defined as a gas phase passive tracer (keyword TRACER in the PROPS section), and initialized to equal the calorific value in units of kJ/sm$^3$ (METRIC), Btu/Mscf (FIELD), J/scc (LAB).

(See "Tracer tracking" in the *ECLIPSE Technical Description* for an account of how to define tracers.)

The keyword is followed by a single record, terminated with a slash (/).

1. Name of the tracer
    (Up to 3 characters)

End data record with a slash (/).

**Example**

```
CALTRAC
CAL / 
```
Component calorific values

This keyword associates a molar calorific value with each component. Either the keyword CALVAL or CALVALR is required if calorific value production rate constraints are to be set for wells (see keyword WCONPROD) or groups (see keyword GCONPROD). It is also required in a GASWAT run if a gas quality calculation is to be performed. (See "GASWAT option" in the ECLIPSE Technical Description for further information.)

The keyword should be followed by \(N_c\) values.

UNITs: kJ/kg-M (METRIC), Btu/lb-M (FIELD), J/gm-M (LAB), kJ/kg-M (PVT-M)

Example

```
---Calorific values in kJ/kg-M
CALVAL
10 20 0.0 0.0 /
```
CALVALR  Component calorific values for multiple equation of state regions

This keyword associates a molar calorific value with each component in each equation of state region. Equation of state regions are defined using the EOSNUM keyword. The CALVALR keyword enables the calorific value production rate for wells completed in multiple equations of state regions to be computed. Note, if crossflow occurs in a production well the calorific value of the injected fluid will be taken as a wellbore or segment average, computed per component as a flow rate weighted average of the producing completions. For injecting wells the calorific values are those of the equation of state associated with the wellbore.

Either of the keywords CALVAL or CALVALR is required if calorific value production rate constraints are to be set for wells (see keyword WCONPROD) or groups (see keyword GCONPROD). It is also required in a GASWAT run if a gas quality calculation is to be performed. (See "GASWAT option" in the ECLIPSE Technical Description for further information.)

The keyword should be followed by \( N_c \) values for each equation of state region.

UNITS: kJ/kg-M (METRIC), Btu/lb-M (FIELD), J/gm-M (LAB), kJ/kg-M (PVT-M)

Example

```plaintext
---Calorific values in kJ/kg-M
CALVALR
10 20 0.0 0.0 /
12 22 0.0 0.0 /
```
CARFIN

Specifies a Cartesian local grid refinement

The CARFIN keyword is used to set up a Cartesian local grid refinement. It specifies a cell or a box of cells identified by its global grid coordinates I1-I2, J1-J2, K1-K2 to be replaced by refined cells. The dimensions of the refined grid within this box are specified as NX, NY, NZ.

Local grid refinement is described in "Local grid refinement and coarsening" in the ECLIPSE Technical Description.

The CARFIN keyword is followed by the following items of data, terminated by a slash (/):

1. Name of the local grid refinement (up to 8 characters).
2. I1 Lower I-coordinate of the box in the parent grid.
3. I2 Upper I-coordinate of the box in the parent grid.
8. NX Number of refined cells along the X-direction.
9. NY Number of refined cells along the Y-direction.
10. NZ Number of refined cells along the Z-direction.
11. NWMAX Maximum number of wells this local refined grid will contain.

ECLIPSE 100 only

12. Name of parent LGR - up to 8 characters. This item may be set to either a null string or to the string ‘GLOBAL’ to indicate that the parent grid is global (that is, this is not a nested refinement).

Nested refinements may be used in ECLIPSE 300 and ECLIPSE 100. If the parent grid is already an LGR then the name of the parent LGR should be specified. The range of I-, J- and K- indices should then refer to the parent grid. If the parent grid is the global grid, then the range of I-, J- and K- indices refers to the global grid.

ECLIPSE 100

Nested refinements cannot be used in ECLIPSE 100 if the parent or child is part of an amalgamated local grid. They also cannot be used with the Parallel or Gradient options.

DEFAULT: ‘GLOBAL’

Notes

• In ECLIPSE 100, and when using the WARP linear solver in ECLIPSE 300, LGRs can only be positioned at a boundary of their parent grid if that boundary is also the boundary of the global grid.

• When the grid is a dual or (in ECLIPSE 300) a multi-porosity grid, then the box of the parent grid is specified in the matrix only. The number of refined cells in the Z-direction is specified in the same way as for the DIMENS keyword in RUNSPEC - namely that NZ (item 10) gives the total number of Z-direction cells in all porosities, so that NZ must be a multiple of the number of porosities. An error will be posted if this is not the case. For example, for a dual porosity case, Z should be a multiple of 2 and NZ /2 should give the number of Z-direction refined cells in the matrix.
Examples

Example 1

CARFIN
--Name  I1  I2  J1  J2  K1  K2  NX  NY  NZ
REF1  19  19  3   3  1  2    3   1    6  /
--Enter data for this local grid
PORO
0.25 0.28 0.21 0.225 0.23 0.18
0.17 0.15 0.22 0.165 0.17 0.16
0.20 0.21 0.19 0.120 0.15 0.17 /
ENDFIN

Example 2

Nested refinement, located at the boundary (K1=1) of both REF1 and the global grid.

--Define a local grid REF1
CARFIN
--Name  I1  I2  J1  J2  K1  K2  NX  NY  NZ  NWMAX  PARENT
REF1  16  19  3   3  1  4   8  4  16  1*  GLOBAL /
ENDFIN
--Define a local grid REF2 contained within REF1
CARFIN
--Name  I1  I2  J1  J2  K1  K2  NX  NY  NZ  NWMAX  PARENT
REF2  2   3   2   3   1   8   4   4   16  1*   REF1 /
--Enter data for this local grid here if required
ENDFIN
Specify Cartesian geometry

The keyword indicates Cartesian (X, Y, Z), rather than radial geometry is to be used for the simulation. Note that as this is the default geometry convention, it is not normally necessary to specify this keyword.

The CART keyword takes no associated data.
Component type for Asphaltene model

This keyword can be used with the Asphaltene option (ASPHALTE) to identify special components in the asphaltene model. It is an alternative to the ASPFLOC keyword. The keyword should be followed by one value for each component. The value should be one of the following:

- **NONE** for an ordinary component
- **PREC** for a component that can precipitate as asphaltene
- **FLOC** for the component that represents the flocculated asphaltene
- **DEPO** for the component that represents deposited asphaltene

**DEFAULT:** NONE

The following restrictions apply when using this keyword:

- Exactly one component should be identified as the precipitate.
- Exactly one component should be identified as the flocculate.
- If the asphaltene deposition model (ASPDEPO) is not used, then there should be no deposition component.
- If the three component asphaltene deposition model is required, then exactly one component should be identified as deposited asphaltene. This component must also be a solid component (using keywords SOLID and CVTYPE).
- If the pre-2011.1 asphaltene deposition model is required, then there should be no deposition component.

If this keyword is used the ASPFLOC keyword will be ignored.

**Example**

```
CATYPE
3* PREC FLOC DEPO/

CVTYPE
5* SOLID
```
CBMOPTS  Options for the Coal Bed Methane Model

The keyword is used to control options associated with the Coal Bed Methane Model. A Coal Bed Methane Model must be activated by including the COAL keyword in the RUNSPEC section.

The CBMOPTS keyword is optional and should be followed by some or all of the following data items, terminated by a slash (/).

1. The adsorption model used. (ECLIPSE 300 only.)
   
   The TIMEDEP model requires that the run is a DUALPORO (or multi-porosity) run while the INSTANT model can also be used for single porosity models.
   
   The model is either INSTANT or TIMEDEP.
   
   DEFAULT: TIMEDEP

2. Specify if water flow in/out of a coal cell is allowed. (ECLIPSE 300 only.)
   
   A coal cell is a cell with a non-zero coal region number as set by the COALNUM keyword. This will only have an effect on runs using the INSTANT adsorption model.
   
   The value is either YES or NO.
   
   DEFAULT: YES

3. Specify if interpolation of oil gas relative permeabilities is allowed. (ECLIPSE 300 only.)
   
   The value is either NOKRMIX or KRMIX.
   
   DEFAULT: NOKRMIX which corresponds to using the keyword NOMIX.

4. Specify that the passes through the rebound region, when the Palmer-Mansoori rock model is used (ROCKPAMA or ROCKPAME), are counted.
   
   The rebound region is where the pressure derivative of the pore volume model changes sign. Warnings are issued for the first detections. Note that this includes passes done within each Newton iteration. The total number of passes through the rebound region can be viewed by the array NPMREB output by RPTSCHED or RPTRST.
   
   NOPMREB or PMREB
   
   DEFAULT: NOPMREB.

5. Specify the method to be used for the Palmer-Mansoori rock model (ROCKPAMA):
   
   PMSTD  Invoke the standard Palmer-Mansoori method.
   
   PMADTN  Use the adsorbed gas in the matrix for the sorption-induced swelling/shrinkage strain term. The gas concentrations at the current timestep, given by the Newton iteration, is used. However, derivatives with respect to the gas concentrations are not included in the Jacobian matrix. For ECLIPSE 300 this method requires TPAMEPS.
   
   PMADIMP  Use the adsorbed gas in the matrix for the sorption-induced swelling/shrinkage strain term and solve for this implicitly. The derivatives with respect to the matrix gas concentrations are included in the Jacobian matrix during the Newton iterations. For ECLIPSE 300 this method requires TPAMEPS.
PMADTT Use the adsorbed gas in the matrix, at the previous timestep, for the sorption-induced swelling/shrinkage strain term. For ECLIPSE 300 this method requires TPAMEPS.

Note that PMADTN, PMADIMP or PMADTT are compatible with using the keyword TPAMEPS and TPAMEPSS.

DEFAULT: PMSTD.

Pre-2013.1 calculations can be obtained by using item 295 in OPTIONS3 for ECLIPSE 300, or item 200 in OPTIONS for ECLIPSE 100.

6. Use the Langmuir scaling value for the volumetric strain term in the Palmer-Mansoori model (ROCKPAMA or ROCKPAME).

NOPMSCAL or PMSCAL

DEFAULT: PMSCAL

7. Select if the Palmer Mansoori model should be applied to the pore volume and/or to the permeability. This enables a subset of the model to be activated (ROCKPAMA or ROCKPAME).

PMPVK Both pore volume and permeability

PMCPVK Compressibility term for pore volume, but full method for permeability

PMK Only for the permeability

PMPV Only for the pore volume

DEFAULT: PMPVK

ECLIPSE 100 only 8. Pressure chop if passing through the rebound pressure.

PMPCHP or NOPMPCHP

DEFAULT: NOPMPCHP


DEFAULT: 5.0D-6

Example

```
CBMÖPTS
    -- model water-flow
    INSTANT NO/
```
**Component compressibility type**

This keyword can be used with the Thermal option to specify the formula used for the liquid phase compressibility of a hydrocarbon component.

By default the volume of a component in the liquid phase is given by:

\[
V_c = \left[1 - c_P \left(\frac{P - P_{ref}}{P_{ref}}\right)\right] \cdot \left[1 + C_T 1_c \left(\frac{T - T_{ref}}{T_{ref}}\right)\right] \cdot \frac{MW_c}{\rho_{ref}}
\]

Eq. 3.12

where the constants used in this equation are described further in keywords `CREF`, `DREF`, `PREF`, `TREF` or `THERMEX1`.

However, when modeling foamy oil, it is convenient to model the gas trapped in the oil as a component in the liquid phase, but with a gas-like compressibility. Keyword `CCTYPE` allows the component liquid phase volume to be modeled using a gas law:

\[
V_c = Z_c RT / P
\]

Eq. 3.13

See keyword `ZFACTOR` for more details.

The keyword should be followed by one value for each component. The value should be one of the following:

- **OIL** for standard liquid phase compressibility shown in equation 3.12
- **GAS** for gas-like liquid phase compressibility shown in equation 3.13

These values may be enclosed in quotes. Only the first character is significant.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword `TABDIMS`. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

**Example**

```
CCTYPE
OIL OIL GAS /
```
CCTYPES

Component compressibility type for surface conditions

This keyword can be used with the THERMAL option to specify the component oil phase compressibility type (OIL or GAS) at surface conditions. If this keyword is not used, the surface compressibility type will be the same as the reservoir compressibility type, set using the CCTYPE keyword.

The keyword is used in the same way as the CCTYPE keyword.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.
CECON

Economic limits for production well connections

CECON is used to set economic limits for production well connections. If an individual connection violates one of its economic limits it will be automatically closed. The well connections must already be defined before this keyword can be used.

CECON is followed by any number of records, described below, each record terminated by a slash (/). A record may be terminated early at any item; the remaining items assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

Each record can contain some or all of the following items:

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. I - location of connecting grid block(s)
   DEFAULT: 0 (allows any I-value)

3. J - location of connecting grid block(s)
   DEFAULT: 0 (allows any I-value)

4. K - location of upper connecting block in this set of data
   DEFAULT: (corresponds to top connection of well)

5. K - location of lower connecting block in this set of data
   DEFAULT: (corresponds to bottom connection of well)

   The following economic limits will be applied to all the connections in the well that match the location indices specified in items 2 - 5. But if a location index is defaulted to zero, it plays no part in selecting which connections have these limits. Thus if the I and J location indices are defaulted to zero, the economic limits are applied to all connections in the well that are located between layers K1 and K2 specified in items 4 and 5. If all four location indices are defaulted, the limits are applied to all the connections in the well.

6. Maximum water cut (water-liquid ratio). User defined arguments can be set with keywords UDQ and UDADIMS.
   A value of 0.0 switches off this limit
   UNITS: sm$^3$/sm$^3$ (METRIC), stb/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)
   DEFAULT: No limit

7. Maximum gas-oil ratio. User defined arguments can be set with keywords UDQ and UDADIMS.
   A value of 0.0 switches off this limit
   UNITS: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)
   DEFAULT: No limit

Keywords
8. Maximum water-gas ratio. User defined arguments can be set with keywords UDQ and UDADIMS. A value of 0.0 switches off this limit.

UNITS: sm$^3$/sm$^3$ (METRIC), stb/Mscf (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)

DEFAULT: No limit

9. Workover procedure when a limit is violated

  CON  Shut the connection

  +CON  Shut the connection and all below it (see keyword COMPORD)

  WELL Shut or stop the well (Item 9 in keyword WELSPECS)

  ECLIPSE 100  PLUG Plug back the well (see keyword WPLUG)

DEFAULT: CON

10. Continue checking stopped wells?

    YES  The connection economic limits continue to be checked even if the well is stopped (as crossflow may still occur).

    NO   Connection economic limits are not checked in stopped wells.

    DEFAULT: NO

11. Minimum oil production rate. User defined arguments can be set with keywords UDQ and UDADIMS.

If its oil production rate falls below this value, the connection is shut (and all below it, if item 9 requests).

UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: - infinity (-1.0E20)

12. Minimum gas production rate. User defined arguments can be set with keywords UDQ and UDADIMS.

If its gas production rate falls below this value, the connection is shut (and all below it, if item 9 requests).

UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: - infinity (-1.0E20)

13. Name of well to be opened when this well is shut (the follow-on well)

If a well name is entered here, the named well is opened when the well specified in item 1 is closed automatically for any reason (for example on violating an economic limit). The named well must be fully specified as a SHUT producer or injector. Manual closure of the well in item 1 (for example using WCONFPROD or WEOCON) does not cause the named well to be opened.

This option can be used, for example, to open an injector when a producer closes after becoming uneconomic. If the injector has the same completions as the producer, the effect is equivalent to converting the producer into an injector, although in this case they are regarded as two separate wells.
If the well to be opened is given a non-zero drilling time in keyword WDRILTIM, the opening is subject to constraints on drilling rig availability (see keyword GRUPRIG).

DEFAULT: '

Notes

- If keyword COMPLUMP is used to lump connections into completions, the connection economic limits are applied to whole completions (that is a whole completion is closed if its total water cut, GOR or WGR exceeds the limit set for its connections, or if its total oil or gas production rate falls below the minimum value set for its connections). All connections in the same completion must be given the same limits.

- Crossflowing (injecting) connections have a negative production rate. Thus setting item 11 or 12 to a small negative value causes the connection to be shut if it attempts to crossflow. The default of -1.0E20 allows the connection to remain open if it crossflows.

- Only one UDA may be defined for each item per well at any one time. For any particular item, if two UDAs are defined for different connections on the same well then that the corresponding value will be calculated for all UDA-controlled connections using the most recently requested UDA.

See also keyword WLIMTOL and the ECLIPSE 100 keyword WORKLIM.

Example

| CECON     | PROD1 4* 0.7 | Water cut limit for all connections in well 'P' 2* 2 1* 3.5 | GOR limit for layer-2 connections in all wells with names beginning with P |

Keywords

438
CECONT is used to set tracer economic limits for production well connections, including polymer and brine. If an individual connection violates one of its economic limits it will be automatically closed. The well connections must already be defined before this keyword can be used.

CECONT is followed by any number of records, described below, each record terminated by a slash (/). A record may be terminated early at any item; the remaining items assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted. There are two types of record; one describes the connection and the second lists the tracers and their limits.

The set of records must end with a blank record, containing only a slash (/).

Records of the first kind can contain some or all of the following items:

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. I - location of connecting grid block(s)
   DEFAULT: 0 (allows any I-value)

3. J - location of connecting grid block(s)
   DEFAULT: 0 (allows any J-value)

4. K - location of upper connecting block in this set of data
   DEFAULT: 0 (corresponds to top connection of well)

5. K - location of lower connecting block in this set of data
   DEFAULT: 0 (corresponds to bottom connection of well)

   The following economic limits will be applied to all the connections in the well that match the location indices specified in items 2 - 5. However, if a location index is defaulted to zero, it plays no part in selecting which connections have these limits. Thus if the I and J location indices are defaulted to zero, the economic limits are applied to all connections in the well that are located between layers K1 and K2 specified in items 4 and 5. If all four location indices are defaulted, the limits are applied to all the connections in the well.

6. Workover procedure when a limit is violated
   CON  Shut the connection
   +CON  Shut the connection and all below it (see keyword COMPORD)

   DEFAULT: CON

7. Continue checking stopped wells?
YES The connection economic limits continue to be checked even if the well is stopped (as
crossflow may still occur).

NO Connection economic limits are not checked in stopped wells.

DEFAULT: NO

End the record with a slash (/)

A maximum of three subsequent records can follow, for each well connection or range of connections.
These records can contain some or all of the following items.

1. Tracer name (Use PLY for a polymer and BRI for brine)
2. Maximum total tracer rate
   DEFAULT: No limit
3. Maximum total tracer concentration (not available for partitioned tracers)
   DEFAULT: No limit.
   The following are only available for two-phase tracers
4. Maximum free tracer rate
   DEFAULT: No limit
5. Maximum free tracer concentration
   DEFAULT: No limit
6. Maximum solution tracer rate
   DEFAULT: No limit
7. Maximum solution tracer concentration
   DEFAULT: No limit.

End the record with a slash (/).

End the data for this well connection with a blank line containing just a slash(/).

Note that this means that the keyword will end with two blank lines containing just slashes.

Notes

• If keyword COMPLUMP is used to lump connections into completions, the connection economic limits
  are applied to whole completions (that is a whole completion is closed if its total rate or concentration
  exceeds the limit set for its connections). All connections in the same completion must be given the
  same limits.

• Crossflowing (injecting) connections have a negative production rate. Thus setting item 11 or 12 to a
  small negative value causes the connection to be shut if it attempts to crossflow. The default of
  -1.0E20 allows the connection to remain open if it crossflows.

• No units are given for the tracer rates. The rates are those of the carrier flow rate.
• Since the tracers are solved after the main reservoir simulation if it is subsequently discovered that a limit is violated, the timestep is always repeated with the action being enforced from the start of the timestep. Even with doing this it may be that the ‘action’ lags the violation by a timestep.

• This keyword needs to end with two lines containing only slashes. See the examples below.

Examples

Example 1

```plaintext
CECONT
  PROD1 4* CON NO /
  TR1 1000.0 0.5 /
  TR2 2* 1* 0.8 2* /
  TR3 1500.0 0.1 /
 /
  PROD2 5 6 3 3 CON+ NO /
  TR3 100.0 0.05 /
 /
  PROD3 6* /
  TR4 200 5* /
  PLY 1* 0.7 /
 /
```

Example 2

```plaintext
CECONT
  PROD4 4* CON NO /
  TR5 1000.0 0.5 /
  TR1 2* 1* 0.8 2* /
  TR3 1500.0 0.1 /
 /
```

Example 3

```plaintext
CECONT
  PRODUCER 8 9 1 1 CON+ NO /
  BRI 1000.0 0.1 /
 /
```
CFLLIMIT

Stable timestepping

The data consists of some or all of the following items, which control the implicit cell choice within an AIM run or the timestep choice within an IMPES run. The data must be terminated by a slash (/).

1. Turns on (or off) the stable timestepping option.
   - ON  Stable timestepping turned on.
   - OFF  Stable timestepping turned off.

   DEFAULT: OFF

2. The target maximum CFL value for any explicit cell.

   DEFAULT: 1.0

   CAUTION: Whilst many cases will run faster, and with no apparent ill effect, with a maximum CFL greater than 1 this is strongly discouraged unless the user is confident of being able to spot when problems have arisen.

3. In AIM runs only, the CFL value above which cells will be made implicit (subject to item 5)

   DEFAULT: 0.75

4. At the end of the timestep the CFL numbers are recalculated with the new solution. If any cell has a CFL above the value given here the timestep is chopped and repeated.

   DEFAULT: 2.0

5. The fraction of cells that can be made implicit in an AIM run.

   DEFAULT: 0.05 (=5%)

   Note: If the predominant reason given for timestep choice in the run is ‘CFL’ then it is usually the case that the run is more efficient if a larger value is given here, that is, more cells are allowed to be implicit.

6. An integer specifying the number of AIM state evaluations that an implicit cell remains implicit after being set, regardless of the selection criteria.

   This option is only activated when item 5 is greater than 0.

   DEFAULT: 1.0

7. An integer which controls the implicit treatment of the wells.

   0  Wells are not given any special treatment.

   1  All the inner solver direction lines of cells containing the completions of open wells are set to be implicit.

   2  All the cells containing the completions of open wells are set to be implicit.

   DEFAULT: 1
8. Turn off limiting of change due to capillary pressure, that is, the keyword DPCDT is ignored. If set to ‘NO’ then the value given to DPCDT in the RUNSPEC section is honored.

DEFAULT: YES

9. An integer specifying the number of timesteps taken before reevaluating the AIM state.

The AIM state is always reevaluated at the beginning of a new report step.

If this item is negative (and item 5 is greater than zero), the AIM state is always reevaluated if throughput or solution change is limiting the current step.

DEFAULT: 1

Example

In this example up to 5% (item 5) of the reservoir could be made implicit if needed, the run is assumed to be AIM. A more conservative choice has been taken for item 4.

```
CFLLIMIT
  'ON' 2 1.5 0.05 1
```

Keywords

443
Component gas density type

This keyword can be used with the THERMAL option to specify how the gas phase molar density is computed. With the keyword present an equation of state (EoS) can be used. The default equation of state is Peng-Robinson (PR). If the EOS keyword is used, the equations of state are the ones defined in the item in EOS. All of the keyword necessary for the EoS, such as BIC, need to be added in the PROPS section when using the CGDTYPE keyword (even when all items are NO).

The keyword should be followed by one value for each component. The value should be one of the following:

<table>
<thead>
<tr>
<th>Type</th>
<th>Component</th>
</tr>
</thead>
<tbody>
<tr>
<td>NO</td>
<td>Not used. The molar volume for this component is computed as if this keyword were not present.</td>
</tr>
<tr>
<td>EMIX</td>
<td>An equation of state is used for the EMIX components. The vapor root of the cubic equation of state is always used.</td>
</tr>
<tr>
<td>EGFE</td>
<td>An equation of state is used for the EGFE components. The molar volume is chosen to be the volume with the lowest Gibbs free energy. Thus a liquid molar volume would be used if the stable phase at the given pressure and composition corresponds to a liquid.</td>
</tr>
</tbody>
</table>

Table 3.3: CGDTYPE component values

If the keyword is present the default is EMIX.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

Note: If only a subset of the mixture is modeled by an EoS and many of the heavy components are included, and the light components are not, the mixture could yield a molar volume similar to that of an OIL or give an unphysical molar volume.

For more information see Thermal option in the ECLIPSE Technical Description.

Example

In a thermal run with 4 hydrocarbon components:

```
CGDTYPE
  EMIX EMIX EMIX EMIX /
```

Keywords

ECLIPSE Reference Manual

444
Component gas viscosity type

This keyword can be used with the THERMAL option to specify how the gas viscosity is computed, such as using the Lorentz-Bray-Clark viscosities. With the keyword present an equation of state (EoS) can be used.

The keyword should be followed by one value for each component. The value should be one of the following:

<table>
<thead>
<tr>
<th>Type</th>
<th>Component</th>
</tr>
</thead>
<tbody>
<tr>
<td>NO</td>
<td>Not used. The viscosity for this component is computed as if this keyword was not present.</td>
</tr>
<tr>
<td>LMIX</td>
<td>The LBC viscosity model is applied with these components’ gas phase molar density.</td>
</tr>
<tr>
<td>LEMIX</td>
<td>The LBC viscosity model is applied on the LEMIX components densities where an EoS is used to compute the phase molar density.</td>
</tr>
</tbody>
</table>

Table 3.4: CGVTYPE component values

If the keyword is present the default is LMIX.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

Note: If only a subset of the mixture is modeled by an EoS and many of the heavy components are included, and the light components are not, the mixture could yield a molar volume similar to that of an OIL and consequently the viscosity will be similar to that of an oil.

For more information see the Thermal option" in the ECLIPSE Technical Description.

Example

In a thermal run with 4 hydrocarbon components:

```
CGVTYPE
LMIX LMIX LMIX LMIX /
```
Increase well dimensions

This keyword allows the well dimensioning information, initially entered using WELLDIMS, to be increased in the Schedule section. This might be done if one of the well dimension limits is exceeded, particularly during a “fast restart” run.

The format of the keyword is similar to WELLDIMS. The data consisting of the following integers which describe the dimensions of the well data to be used in the run.

Items may be defaulted, in which case they will retain their original values. No action will be taken if none of the dimensions is increased.

In order to increase the size of the well arrays concerned, the existing arrays are written to a scratch file, the space used compressed out, the arrays reallocated and read back. This is a fairly expensive process, and it is always preferable to set up sufficiently large WELLDIMS dimensions initially. In fact the use of CHANDIMS may be avoided altogether by using “flexible restarts”, which allow the well dimensions to be increased at each restart (see keyword RESTART).

The data items entered are:

1. The maximum number of wells
   DEFAULT: Same as the value set in WELLDIMS
2. The maximum number of completions per well.
   DEFAULT: Same as the value set in WELLDIMS
3. The maximum number of groups.
   DEFAULT: Same as the value set in WELLDIMS
4. The maximum number of wells in a group, or child groups in a group.
   DEFAULT: Same as the value set in WELLDIMS
5. The maximum number of stages per separator including the last, which represents the stock tank.
   DEFAULT: Same as the value set in WELLDIMS
6. The maximum number of wellstreams.
   DEFAULT: Same as the value set in WELLDIMS
7. The maximum number of mixtures.
   DEFAULT: Same as the value set in WELLDIMS
8. The maximum number of separator conditions.
   DEFAULT: Same as the value set in WELLDIMS
9. The maximum number of mixture items.
   DEFAULT: Same as the value set in WELLDIMS
10. Item not used.
11. The maximum number of well lists that a well may belong to at any one time.
    DEFAULT: Same as the value set in WELLDIMS
12. The maximum number of dynamic well lists in the simulation.

    DEFAULT: Same as the value set in WELLDIMS

**CAUTION:** CHANDIMS must not be used to change any well dimensions if the run contains any multisegment wells. Well dimensions may still be changed, however, in flexible restarts using the RESTART keyword.

**Example**

```
    CHANDIMS 20 40 4* 10 4 /
```
CIRCLE

Requests completion of the circle

This keyword requests that non-neighbor connections be generated to complete the circle in radial geometry. These connections act between the last cell in the azimuthal, \( \theta \), direction, and the first cell, for each layer and ring of cells in the R-direction.

Such connections are only generated if the grid subtends 360 degrees, so that the first and last cells are in contact.

This keyword has no items, and only has an effect if the **RADIAL** keyword is specified in the **RUNSPEC** section.

Example

```plaintext
CIRCLE
```
Component names

In a run with $N_c$ components, this keyword associates an identifying character string with each component for reporting purposes. These may be of up to eight characters.

The keyword should be followed by $N_c$ values. These may be enclosed in quotes, although these are only required when blank characters are used, or the name starts with a non-alphabetical character.

Example

```
CNAMES
Meth Ethane C3-C6 C7+ /
```
CO2SOL

Allow CO2 to dissolve in the aqueous phase

This keyword turns on the CO₂ Solution option. The CO₂ component is then allowed to exist in all three phases. This keyword cannot be used if directional saturation table end-point scaling is used (switched on by setting the first item of keyword ENDSCALE to DIRECT).

CO2SOL is the only keyword required for this option. There are several optional keywords which can be used to modify the solubility and define initial conditions.

If the CO2SOL keyword is used on its own, default formulae will be used to create solubility tables. The default formulae depend on the salinity, which is taken to be zero, unless specified by one of the following keywords:

- **SALINITY**: Define the salinity.
- **SALINITR**: Define the salinity on a regional basis.

Alternatively, user specified solubility tables can be entered using one of the following keywords:

- **SOLUBLI**: This allows values in the default solubility table to be entered by the user.
- **SOLUBLI S**: This allows the user to enter a solubility table with salt dependencies.
- **SOLUBLIT**: This allows the user to enter a solubility table with temperature dependencies.

The following keywords can optionally be used to set initial conditions:

- **RSWVD**: This specifies the initial variation of aqueous CO₂ gas-water ratio versus depth.
- **RSW**: This specifies the initial variation of aqueous CO₂ gas-water ratio by enumeration.
- **AMF**: This specifies the initial composition of aqueous CO₂ system.

For further information see "CO₂/H₂S solution in the aqueous phase" in the ECLIPSE Technical Description.
CO2STORE  CO2 storage option

This keyword turns on the CO₂ storage option. The mutual solubility and fluid properties of CO₂ and H₂O are considered, where the H₂O rich phase might include salts.

CO2STORE is the only keyword required for this option.

For further information see "CO₂ storage" in the ECLIPSE Technical Description.
Enable Coal Bed Methane option

This keyword enables the coal bed methane option (see "Coal bed methane model" in the ECLIPSE Technical Description).

The keyword has no associated data.
COALADS  

**Gas/solvent relative adsorption data**

Either the COALADS keyword or the COALPP keyword should be used to specify the relative adsorption data in runs containing solvent. The COALADS keyword tabulates simple relative adsorption factors for the gas (methane) and solvent. See “Coal bed methane model” in the ECLIPSE Technical Description for further information.

The data comprises NTCREG (see keyword REGDIMS, item 6) tables of gas/solvent relative adsorption functions, each terminated by a slash (/).

The tables consist of 3 columns of data. The first column is the phase fraction of gas, $S_{\text{gas}} / (S_{\text{gas}} + S_{\text{solvent}})$. The next two columns are respectively the relative adsorption of gas ($F_g$) and the relative adsorption of solvent ($F_s$).

### Column:

1. The gas phase fraction ($f_g$). Values should span the range 0 to 1 and increase monotonically down the column.
2. The corresponding gas relative adsorption ($F_g$). Values should be between 0 and 1 and should be level or increasing down the column.
3. The corresponding solvent relative adsorption ($F_s$). Values should be between 0 and 1 and should be level or decreasing down the column.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see keyword TABDIMS).

The surface concentration of gas and solvent at a given pressure and composition are given by:

Surface concentration of gas = $C_g (P) \cdot F_g (f_g)$

Surface concentration of solvent = $C_s (P) \cdot F_s (f_g)$

where

- $C_g (P)$ is the adsorption of pure gas at pressure $P$ (LANGMUIR keyword)
- $C_s (P)$ is the adsorption of pure solvent at pressure $P$ (LANGSOLV keyword)
- $F_g$ is the gas relative adsorption
- $F_s$ is the solvent relative adsorption
- $f_g$ is the fraction of gas in the fracture ($= S_{\text{gas}} / (S_{\text{gas}} + S_{\text{solvent}})$)

The coal region number COALNUM is used to look up the table.

If item 180 of the OPTIONS keyword is activated, the number of tables should equal NTSFUN (see keyword TABDIMS) and the region keyword SATNUM is used to look up the table.

### Example

Where NTCREG is 2 and NSSFUN is $\geq 4$.

---

**Keywords**
COALADS
--
<table>
<thead>
<tr>
<th>Fg</th>
<th>Fs</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>1.0</td>
<td>1.0</td>
</tr>
</tbody>
</table>
/
| 0.0| 0.0| 1.0|
| 0.4| 0.2| 0.8|
| 0.8| 0.7| 0.2|
| 1.0| 1.0| 0.0|
/
COALNUM

Coal regions

The **COALNUM** keyword is used with the coal bed methane option to define regions with coal. The region number entered with this keyword is used to determine the sorption of the coal. See keyword **LANGMEXT** (ECLIPSE 300) or keyword **LANGMUIR**. A zero value implies there is no coal and that the standard dual porosity fluid flow equations should be solved. The keyword should be followed by one integer for each grid cell in the matrix. The data entries should not exceed **NTCREG** entered using keyword **REGDIMS** in the **RUNSPEC** section (item 6). The data must be terminated by a slash (/).

Any matrix cell not explicitly assigned a **COALNUM** value will be defaulted to a value of unity (ECLIPSE 300).

If the keyword is not present all matrix cells are assigned a value of unity.

*Note:* If a non-coal region is specified, the usual matrix porosities and permeabilities must be entered for the region.

**Examples**

A 11*11*2 dual porosity grid with a coal matrix. Top layer is matrix, bottom layer is fracture.

```
COALNUM
  121*1
  121*0 /
```

A 4*4*3 single porosity grid with two layers of coal and one layer without coal. This example is based on the use of the E300 instantaneous desorption model.

```
COALNUM
  16*1
  16*2
  16*0 /
```
**COALNUMR  Redefine coal region numbers for matrix subcells**

The **COALNUMR** keyword is used with the Coal Bed Methane option to define variation in the adsorbed gas behavior of subgrid matrix cells when the multi porosity option is used. For more information, see "Multi porosity" in the ECLIPSE Technical Description. This keyword redefines the coal region numbers for subgrid matrix cells. This keyword is optional. By default all subgrid matrix cells will be assigned the coal region number assigned to the grid cell with keyword **COALNUM**. Please note that if **COALNUM** is not present all matrix cells will be assigned coal region number 1 by default.

The keyword requires one record for each **COALNUM** region to be provided with subgrid coal region number variation. Each record starts with the identifier coal region number. The following region numbers represent the ‘new’ coal region number of all matrix cells, first the outermost cell followed by the nested subcell’s coal region numbers. The first entry acts therefore as an identification number followed by **NMATRIX** entries. The maximum number of records is limited by **REGDIMS** item 6. The diffusion data for the coal regions are specified with **DIFFCBM**, while the adsorption characteristics are specified with the keyword **LANGMEXT**.

The first item of each record must be in increasing order.

Default values:

- The first item must not be defaulted and correspond to a coal region number already specified for the outermost primary matrix grid cells.
- The following numbers can be defaulted.
  - The outermost primary matrix cells and intermediate subgrid matrix cells will not be assigned any coal region number and no adsorbed gas is associated with these cells.
  - The inner subgrid matrix cells will be assigned the coal region number as specified in the primary matrix grid cells (same as the first item).

If a record is missing, the innermost subgrid matrix cells will be assigned the coal region number from the primary matrix grid cell and the other subgrid matrix cells will be conventional pore volume cells.

**Note:** Using this keyword with default values alter the coal region numbers so only the innermost subgrid matrix cells are assigned a coal region number.

For further information see "Coal bed methane model" in the ECLIPSE Technical Description.

Each record must be terminated by a slash (/) and the keyword is terminated by a second slash (/).

**Examples**

**Example 1**

This example assumes that the user has specified two matrix cells using the **NMATRIX** keyword and that two coal region numbers are set with **COALNUM**. The primary matrix grid cells with region number 1 are redefined to have no coal region number, the next subgrid matrix cell uses the Langmuir /diffusion data for region number 1. Next record is for cells with region number 2. These are also set to conventional pore volume cells while the inner subgrid matrix cells uses coal region number 3.

```
COALNUMR
1 0 1 /
2 0 3/ 
```

**Keywords**

ECLIPSE Reference Manual

456
Example 2

In this example we assume that the instant desorption model is used, see CBMOP TS. A coal region can then be defined for each of the subgrid matrix cells, while the time dependent method requires coal connections to be with pore volume grid cells or to cells with the same coal region numbers. Recall that for the instant desorption model coal cells also have pore volume.

Coal region number 1 in the primary matrix grid cells is set to 1. For the submatrix grid cells we get coal region number 2. The next record tells that coal region number 2 in the primary matrix grid cells is set to 1, while the following subgrid matrix cells get coal region number 3.

```
COALNUMR
  1 1 2/
  2 1 3/
/
```
COALPP

Gas/solvent partial pressure data

Either the COALPP or the COALADS keyword should be used to specify the relative adsorption data in runs containing solvent. The COALPP keyword tabulates the pressure factor used in a partial pressure model of the relative adsorption. See "Coal bed methane model" in the ECLIPSE Technical Description for further information.

The data comprises NTCREG (see keyword REGDIMS, item 6) tables of gas/solvent relative adsorption functions, each terminated by a slash (/).

The tables consist of 3 columns of data. The first column is the phase fraction of gas, $S_{\text{gas}}/(S_{\text{gas}} + S_{\text{solv}})$. The next two columns are respectively the relative adsorption of gas ($F_g$) and the relative adsorption of solvent ($F_s$).

Column:

1. The gas phase fraction ($f_g$). Values should span the range 0 to 1 and increase monotonically down the column.
2. The corresponding gas relative adsorption ($F_g$). Values should be between 0 and 1 and should be level or increasing down the column.
3. The corresponding solvent relative adsorption ($F_s$). Values should be between 0 and 1 and should be level or decreasing down the column.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see keyword TABDIMS).

The surface concentration of gas and solvent at a given pressure and composition are given by:

Surface concentration of gas = \[ C_g (P \cdot F_g (f_g)) \]

Surface concentration of solvent = \[ C_s (P \cdot F_s (f_g)) \]

where

\[ C_s (P \cdot F_g (f_g)) \] is the adsorption of gas at partial pressure \( P \cdot F_g (f_g) \) (LANGMUIR keyword)

\[ C_s (P \cdot F_s (f_g)) \] is the adsorption of solvent at partial pressure \( P \cdot F_s (f_g) \) (LANGSOLV keyword)

$F_g$ is the gas pressure multiplier

$F_s$ is the solvent pressure multiplier

$f_g$ is the fraction of gas in the fracture ($= S_g / (S_g + S_{\text{solv}})$)

The coal region number COALNUM is used to look up the table.

If item 180 in OPTIONS is activated, the number of tables should equal NSSFUN (see keyword TABDIMS) and the region keyword SATNUM is used to look up the table.

Example

With NTCREG=1 and NSSFUN≥4. Unmodified partial pressure model.

Keywords

458
COALPP
-- Fg Fs
  0.0  0.0  1.0
  1.0  1.0  0.0
/
COARSEN  Specifies a box of cells for grid coarsening

The keyword is followed by any number of records, each containing the following items of data, and each terminated by a slash (/).

Each record describes a box of grid cells for coarsening.

The data items are as follows:

1. Lower I-coordinate of the box (I1)
2. Upper I-coordinate of the box (I2)
3. Lower J-coordinate of the box (J1)
4. Upper J-coordinate of the box (J2)
5. Lower K-coordinate of the box (K1)
6. Upper K-coordinate of the box (K2)
7. Number of coarsened cells along the X-direction in this box (NX)
   - NX must be an exact factor of I2 - I1 +1.
   - DEFAULT: 1
8. Number of coarsened cells along the Y-direction in this box (NY)
   - NY must be an exact factor of J2 - J1 +1.
   - DEFAULT: 1
9. Number of coarsened cells along the Z-direction in this box (NZ)
   - NZ must be an exact factor of K2 - K1 +1.
   - DEFAULT: 1

The set of records must end with a blank record containing just a slash.

The Grid Coarsening option is described in further detail in "Local grid coarsening" in the ECLIPSE Technical Description.

Notes

In addition, the following conditions must be satisfied:

1 ≤ I1 ≤ I2 ≤ NX
1 ≤ J1 ≤ J2 ≤ NY
1 ≤ K1 ≤ K2 ≤ NZ.

The data items specified above divides the box I1 - I2, J1 - J2, K1 - K2 into NX * NY * NZ coarse cells.

Grid coarsening automatically generates non-neighbor connections. Transmissibilities and diffusivities across the cells are modified accordingly. If the NONNC keyword has been entered in RUNSPEC section, an error message is issued, and this option is disallowed.

It is important to note that coarsening boxes must not overlap one another, although they may join together. Furthermore, coarsening boxes should not overlap any local grid refinement.
The parameter specified as the 3rd item (MCOARS) in keyword LGR in the RUNSPEC section must be set equal to or greater than the total number of coarse cells. See also the keyword AUTOCOAR.

**Example**

An example using a 6 x 6 x 1 grid.

```
COARSEN
- I1 I2 J1 J2 K1 K2 NX NY NZ
  1  6  1  2  1  1  3  1  1   /
  1  2  3  4  1  1  1  1  1   /
  5  6  3  4  1  1  1  1  1   /
  1  6  5  6  1  1  3  1  1   /
 /
```
Component oil density type

This keyword can be used with the THERMAL option to specify how the oil phase molar density is computed. With the keyword present an equation of state (EoS) can be used. The default equation of state is Peng-Robinson (PR). If the EOS keyword is used, the equations of state are the ones defined in the item of EOS. All of the keyword necessary for the EoS, such as BIC, need to be added in the PROPS section when using the CODTYPE keyword (even when all items are NO).

The keyword should be followed by one value for each component. The value should be one of the following:

<table>
<thead>
<tr>
<th>Type</th>
<th>Component</th>
</tr>
</thead>
<tbody>
<tr>
<td>NO</td>
<td>Not used. The molar volume for this component is computed as if this keyword was not present.</td>
</tr>
<tr>
<td>EMIX</td>
<td>An equation of state is used for the EMIX components. The liquid root of the cubic equation of state is always used.</td>
</tr>
<tr>
<td>EGFE</td>
<td>An equation of state is used for the EGFE components. The molar volume is chosen to be the volume with the lowest Gibbs free energy. Thus a vapor molar volume would be used if the stable phase at the given pressure and composition corresponds to a vapor.</td>
</tr>
</tbody>
</table>

Table 3.5: CODTYPE component values

If the keyword is present the default is EMIX.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

Note: If only a subset of the mixture is modeled by an EoS and many of the light components are included, and the heavy components are not, the mixture could yield a molar volume similar to that of a GAS.

For more information see Thermal option in the ECLIPSE Technical Description.

Example

In a thermal run with 4 hydrocarbon components:

```plaintext
CODTYPE
  EMIX  EMIX  EMIX  EMIX  /
```

Keywords

462
COLLAPSE identifies collapsible cells in the compressed VE option

The COLLAPSE keyword may optionally be used in compressed VE runs (keyword VE in the RUNSPEC section). It specifies which cells can be collapsed, or merged with other cells in the same column, to form the larger compressed VE cells.

The keyword should be followed by one integer for each grid cell in the model. The integers should be equal to either 0 or 1:

0  Implies that a cell will not be merged with its lower neighbor.
1  Implies that a cell is available to be merged with its lower neighbor.

DEFAULT: 1 (throughout the grid.)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*1). Note that spaces must not be inserted on either side of the asterisk.

A cell merges with the next active cell below, provided that none of the following situations occur:

- COLLAPSE = 0 for that cell
- TRANZ = 0.0
- PERMZ ≤ critical value entered in CRITPERM, if this keyword is present
- There is an inactive cell with DZ > 0.0 between the two active cells
- Either cell has been refined using RADFIN or CARFIN (see "Local grid refinement and coarsening" in the ECLIPSE Technical Description).

See also keyword CRITPERM and "Compressed vertical equilibrium option" in the ECLIPSE Technical Description.

Examples

Example 1

To keep the top layer as original cells. (NDIVIX =10, NDIVIY =10 and NDIVIZ =4 as specified in RUNSPEC keyword DIMENS.)

```
COLLAPSE
100*0
100*1
100*1
100*1 /
```

Example 2

To return the model to a full 3D VE model. (NDIVIX =10, NDIVIY =10, NDIVIZ =4)

```
COLLAPSE
400*0 /
```
COLUMNS  Sets the left and right margins for the input data file

Sets the left and right margins of an input data file. Only the data within the margins and at the margins is read and processed by the free format input parser.

The keyword should be followed by a line containing two integers (the left and right margin settings respectively), terminated with a slash (/). Their values must lie between 1 and 132, and the right margin setting must be greater than the left margin setting.

The default settings of the margins are 1 and 132 for the left and right margins respectively.

Suppose an input data file contained line numbers on the right side at columns 72-80, by setting the right margin to 71 these line numbers are ignored and not read in. Also an input data file may contain a carriage control character at column 1, then to prevent reading in column 1 the user may set the left margin to 2.

Example

```
COLUMNS
  1 71 / Sets left margin at 1 and right margin at 71
COLUMNS
  2 80 / Sets left margin at 2 and right margin at 80
```
COMPAGH  

**Completion gravity head**

COMPAGH is used to set the gravity head for a completion. The pressure at each completion, $P_{\text{comp}}$, is calculated by:

$$P_{\text{comp}} = P_{\text{BH}} + AGH \cdot (D_{\text{comp}} - D_{\text{BH}})$$

Eq. 3.14

where:

$P_{\text{BH}}$  Pressure at bottom hole reference depth

$AGH$  Average gravity head between $D_{\text{comp}}$ and $D_{\text{BH}}$

$$AGH = \left(\frac{\left[\int_{D_{\text{comp}}}^{D_{\text{BH}}} \rho (\delta) g d\delta\right]}{\left[\int_{D_{\text{comp}}}^{D_{\text{BH}}} d\delta\right]}\right)$$

Eq. 3.15

$D_{\text{comp}}$  Depth of completion

$D_{\text{BH}}$  Depth at which bottom hole pressure is defined

Each completion’s pressure head is calculated using the average gravity head value specified in this keyword and the difference in depth between the completion and the reference depth. The average gravity head values specified here overrides those values calculated internally by ECLIPSE 300, based on completion inflows (see item 12 of keyword **WELSPECS**).

The keyword is followed by any number of records, each containing the items of data described below, and each terminated by a / character.

The records can be terminated early using the / character, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. **Well name, well name template, well list or well list template**
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword **WLIST**.

2. **I - location of connecting grid block(s)**
   
   DEFAULT: 0 (allows any I-value)

3. **J - location of connecting grid block(s)**
   
   DEFAULT: 0 (allows any J-value)

4. **K - location of upper connecting block in this set of data**
   
   DEFAULT: 0 (corresponds to top connection of well)

5. **K - location of lower connecting block in this set of data**
   
   DEFAULT: 0 (corresponds bottom top connection of well)
6. *AGH*, the average gravity head

   UNITS: bars/m *(METRIC)*, psi/ft *(FIELD)*, atm/cm *(LAB)*, atm/m *(PVT-M)*

   DEFAULT: 0.0

**Example**

```
COMPAGH
WELL1 18 6 8 23 0.04 /
```
**COMPAGHL  Completion gravity head in local refined grids**

COMPAGHL is used to set the gravity head for a completion in a local refined grid (LGR) cell. The keyword data is similar to that for COMPAGH, except that there is an additional item at position 2 which gives the name of the LGR in which the completion is located.

The pressure at each completion, \(P_{\text{comp}}\), is calculated by:

\[
P_{\text{comp}} = P_{\text{BH}} + AGH \cdot (D_{\text{comp}} - D_{\text{BH}})
\]

where

- \(P_{\text{BH}}\) Pressure at bottom hole reference depth
- \(AGH\) Average gravity head between \(D_{\text{comp}}\) and \(D_{\text{BH}}\),

\[
AGH = \frac{\int_{D_{\text{comp}}}^{D_{\text{BH}}} \rho (\delta) g d\delta}{\int_{D_{\text{comp}}}^{D_{\text{BH}}} d\delta}
\]

Each completion’s pressure head is calculated using the average gravity head value specified in this keyword and the difference in depth between the completion and the reference depth. The average gravity head values specified here will override those values calculated internally by ECLIPSE 300, based on completion inflows (see item 13 of keyword WELSPEC).

The keyword is followed by any number of records, each containing the items of data described below, and each terminated by a / character.

The records can be terminated early using the / character, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Well name, well name template, well list or well list template
   - A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Local grid name

3. I - location of connecting grid block(s) within the local grid
   - DEFAULT: 0 (allows any I-value)

4. J - location of connecting grid block(s) within the local grid
   - DEFAULT: 0 (allows any J-value)

5. K - location of upper connecting block within the local grid in this set of data
   - DEFAULT: 0 (corresponds to top connection of well)
6. K - location of lower connecting block within the local grid in this set of data
   DEFAULT: 0 (corresponds to bottom connection of well)

7. \( AGH \), the average gravity head
   UNITS: bars/m (METRIC), psi/ft (FIELD), atm/cm (LAB), atm/m (PVT-M)
   DEFAULT: 0.0

**Example**

```
COMPAGHL
WELL1 LGR1 3 3 1 4 0.04 /
/```

**Keywords**

468
**COMPDAT** Well completion specification data

**COMPDAT** specifies the position and properties of one or more well completions. This must be entered after the **WELSPECS** keyword defining the appropriate well.

The keyword is followed by any number of records, described below, each record terminated by a slash (/). A record may be terminated at any item after item 9; the remaining items will assume their default values. If the well bore diameter is not required (see item 9), the record can be terminated after item 8.

A single record can be used to assign the properties of several connections of a well, as long as they are situated in the same vertical column of grid blocks. A deviated well can be completed in more than one column, but each column requires a separate record.

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the **ECLIPSE Technical Description** for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword **WLIST**.

2. I - location of connecting grid block(s)

   If set to 0 or defaulted, the I - location of the well head (entered in keyword **WELSPECS**) will be used.

3. J - location of connecting grid block(s)

   If set to 0 or defaulted, the J - location of the well head (entered in keyword **WELSPECS**) will be used.

4. K - location of upper connecting block in this set of data

5. K - location of lower connecting block in this set of data

   The following data then applies to each layer (or connection) with K - locations between the upper and lower values given above.

6. Open/shut flag of connection

   **OPEN** Connection open to flow

   **SHUT** Connection closed off

   **ECLIPSE 100 only**

   **AUTO** Connection initially closed, but will be opened automatically when another connection in the well is closed during an automatic workover. A connection on **AUTO** is opened each time the well is worked over, in the order in which the connections are first defined in **COMPDAT**. Connections on **AUTO** are also opened when necessary to maintain group production rate targets if the **REPERF** action is selected in keyword **PRORDER**.

   **ECLIPSE 300 Thermal**

   **HEAT** Connection is defined to model heat transfer to the formation when using the multisegment well model. Heat connections cannot be opened to fluid flow during the simulation.

   **ECLIPSE 300 only**

   In pre-2012.1 releases, any input other than **SHUT** would result in an **OPEN** connection. This behavior has been changed in subsequent releases so that only **OPEN**, **SHUT** or **HEAT** can be entered.

   **DEFAULT**: **OPEN**

7. Saturation table number for connection relative permeabilities

---

**Keywords**

469
If a default, zero or negative value is entered, the relative permeabilities are calculated using the same saturation table as the grid block containing the connection, selected with keyword SATNUM.

In runs using the Hysteresis option (keyword SATOPTS in RUNSPEC), the saturation table number entered here apply to both imbibition and drainage processes. If a different table number is required for imbibition, it should be set using COMPIMB after the COMPDAT keyword. If a default, zero or negative value is entered, the respective tables for the grid block containing the connection (defined using SATNUM for the drainage table and IMBNUM for the imbibition table) is used.

**ECLIPSE 100**

In runs using the Vertical Equilibrium option (keyword VE in RUNSPEC) the end-points of the saturation table are used to calculate the contact depths and relative permeabilities at the well connection.

**ECLIPSE 300**

If the ODD3P relative permeability and capillary pressure hysteresis model has been selected and a non-default saturation table number is specified, this will be ignored.

8. Transmissibility factor for the connection

If defaulted or set to zero, the connection transmissibility factor is calculated using the remaining items of data in this record. See “The connection transmissibility factor” in the ECLIPSE Technical Description for an account of the methods used in Cartesian and radial geometries. The well bore diameter must be set in item 9.

**UNITS:** cP.rm$^3$/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm$^3$/day/atm (PVT-M)

- If the well is situated on an edge (or a corner) of a Cartesian grid, for example in a sector model or symmetry element, use keyword WPIMULT after COMPDAT to scale the resulting connection factors by 0.5 (or 0.25).

- A well connection in an outer-most block of a radial grid (I = NDIVIX, which is specified in RUNSPEC keyword DIMENS) is assumed to occupy the entire outer boundary of the block for the purpose of applying boundary conditions. If you wish to model a real well in an outer block of a radial grid, the calculated connection factor will be much too large. In this case, you should either supply an appropriate connection factor value yourself, or add an extra ring of inactive cells so that the well is no longer in an outer block.

If a value > 0.0 is entered, the connection transmissibility factor is set to this value. The value will also be used to calculate either the connection’s effective Kh or the denominator term \( \ln(r_o/r_w) + S \) according to the following rules:

- If the Kh value in item 10 is defaulted or set negative, and the well bore diameter set in item 9, the Kh value is calculated from the connection transmissibility factor and the denominator term. In this case, however, the Kh is only used for output (if Schedule data output is requested in RPTSCHED). The denominator term will be calculated from the grid block dimensions, as described in the “The connection transmissibility factor” in the ECLIPSE Technical Description.

- If the Kh value is set > 0.0, the denominator term is calculated from the connection transmissibility factor and the given Kh value, instead of the well bore radius and the grid block dimensions.

- If the Kh value is set = 0.0, it is replaced by the effective Kh of the grid block, then used to calculate the denominator term in the same way as if it had been set > 0.0.

- In ECLIPSE 100, the recalculation of Kh or the denominator term may be turned off by setting item 45 in the OPTIONS keyword > 0. The denominator term is then always calculated from the
grid block dimensions and the well bore radius. This option can be used to achieve back-
compatibility with pre-95A datasets, if required.

The denominator term is used when calculating the well's productivity/injectivity index and when
calculating the effects of a D-factor (see "Well Inflow Performance" in the ECLIPSE Technical
Description). Thus, when the connection factor is set to a positive value, the results of these
calculations depend on the setting of the Kh value in item 10.

**Note:** If a positive value is entered for the connection transmissibility factor, this should include the
effect of any skin entered in item 11, but exclude the effect of any D-factor. The effect of a D-factor is
modeled separately by adjusting the mobility (see "The flow-dependent skin factor" in the ECLIPSE
Technical Description).

9. Well bore diameter at the connection

This quantity is required for:

- Calculating the connection transmissibility factor
- (If a zero or negative value was supplied for item 8)
- Calculating the well’s productivity/injectivity index
- (If a positive value was supplied for the drainage radius in item 7 of keyword WELSPECS)
- Calculating the effects of a D-factor

The quantity is ignored if not required.

**UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

**ECLIPSE 300**

DEFAULT: 1 ft = 0.3048 m

**ECLIPSE 100**

DEFAULT: Undefined.

10. Effective Kh (permeability x thickness) value of the connection

If a default, zero or negative value is entered, the Kh value is calculated from the grid block data.

If a positive value for the connection transmissibility factor is entered in item 8, the calculation of the
denominator term depends on whether the Kh value is set or defaulted. See item 8.

This quantity is required for calculating the connection transmissibility factor (if a default or zero
value was supplied for item 8). It is ignored if not required.

**UNITS:** mD.m (METRIC), mD.ft (FIELD), mD.cm (LAB), mD.m (PVT-M)

DEFAULT: Negative

11. Skin factor

This quantity is used for the same purposes as listed for item 9. It is ignored if not required.

Large negative skin factors can cause problems if they increase the effective wellbore radius to
approach the pressure equivalent radius of the grid block. Very high values of the connection
transmissibility factor may result, which could cause convergence problems. In these circumstances, it
may be preferable to change the grid block data near the well to model the physical cause of the
negative skin (for example enhance the permeability out to a certain distance from the well). If the
effective wellbore radius is increased beyond the pressure equivalent radius, the connection factor
becomes negative and an error message is issued.
ECLIPSE 100

A warning message will be issued if the effective wellbore radius increases beyond half the pressure equivalent radius, or extends outside the grid block.

DEFAULT: 0.0

12. D-factor, for handling the effects of non-Darcy flow of free gas

A D-factor measured for the well should preferably be entered with keyword WDFAC, and this item should be left defaulted. But it may be entered here instead, for back-compatibility with old ECLIPSE 100 datasets. ECLIPSE converts the well D-factor into D-factors for the connections. See the WDFAC keyword for information on how the conversion is performed. Note that the well D-factor must reflect the current number of open connections at the time it is specified.

If you wish to enter the connection’s D-factor directly, you can enter it here with a negative sign. A negative sign tells ECLIPSE to regard it as the connection’s D-factor rather than the well’s D-factor. ECLIPSE simply reverses its sign without applying any conversion.

UNITs: day/sm^3 (METRIC), day/Mscf (FIELD), hr/scf (LAB), day/sm^3 (PVT-M)

DEFAULT: “Keep the current value of the connection’s D-factor”

13. Direction in which the well penetrates the grid block

X the well penetrates the grid block in the X direction

Y the well penetrates the grid block in the Y direction

Z the well penetrates the grid block in the Z direction

ECLIPSE 300 only

FX the well is completed in a fracture in the X direction

FY the well is completed in a fracture in the Y direction

DEFAULT: Z

In radial grids, only penetration in the Z direction is allowed.

ECLIPSE 300

For Cartesian grids in ECLIPSE 300, if a direction other than Z is specified, the HWELLS keyword must be included in the RUNSPEC section. If FX or FY is specified, the keyword FWELLS must be included in the RUNSPEC section.

This item is used in calculating the transmissibility factor of the connection if it was not supplied in item 8. In Cartesian grids the Peaceman formula is used, taking the components of permeability and grid block dimensions appropriate to the direction of penetration (see "The connection transmissibility factor" in the ECLIPSE Technical Description). Note that the formula assumes that the well completely penetrates a rectangular block through its center in a direction perpendicular to two of its faces.

This item also influences the way ECLIPSE orders the connections in the well (except for friction and multisegment wells) if the default TRACK method is used. See keyword COMPORD for further details.

14. Pressure equivalent radius, r_o

If a value > 0.0 is entered, r_o is set to this value instead of being calculated from the grid block dimensions according to Peaceman’s formula (see "The connection transmissibility factor" in the ECLIPSE Technical Description). Note that the pressure equivalent radius is not used for connections in the innermost or outermost cells of a radial grid or, in ECLIPSE 300, for wells completed in fractured cells.
UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: Calculate \( r_0 \) from Peaceman’s formula

Notes

**ECLIPSE 100**
- The calculated values of the connection transmissibility factor and \( K_h \) (if defaulted) can be examined by requesting a printout of the SCHEDULE section keyword data (argument WELSPECS in keyword RPTSCHED). The connection transmissibility factor can also be written to the SUMMARY file with the SUMMARY section keyword CTFAC.

- If you wish to prevent inter-block flow (for example due to shale breaks or faults), use keywords MULTZ and MULTX for example to set a zero transmissibility multiplier. Don’t set the rock permeability to zero instead, as it is needed to calculate the connection transmissibility factor.

- There is a choice of methods for determining the ordering of the connections along the wellbore. See keyword COMPORD for further details.

**ECLIPSE 300**
- If non-Darcy Forchheimer effects for gas flow have been enabled using item 4 of the VELDEP keyword, then a D-factor for each production well connection will be calculated. These calculated, Forchheimer parameter based D-Factors will override any set in this keyword. See “Forchheimer parameter based well D-factors” in the ECLIPSE Technical Description for further details.

Example

```
COMPDAT
PROD1 6 8 1 3 OPEN 0 0.0 0.333 500 3.5 1E-3 Z /
PROD2 0 0 3 3 OPEN 3 0.0 0.333 /
PROD3 3 5 2 2 OPEN 1* 0.333 3* X /
PROD3 4 5 2 3 OPEN 1* 0.333 3* X /
*INJ* 0 0 3 3 OPEN 0 2.3 /
```

Keywords

473
Completion data for wells in local grids

COMPDATL must be used in place of COMPDAT to specify the connection data for wells in local refined grids, after the wells have been introduced with keyword WELSPECL.

The keyword data for COMPDATL is the same as for COMPDAT, except for an extra item (item 2) which names the local grid containing the connections specified in the record. The well may be completed in more than one local grid, but in ECLIPSE 100 these local grids must be amalgamated using the keyword AMALGAM.

Note: In ECLIPSE 100, the format of this keyword was altered in the 2000A release, with the addition of the local grid name in item 2. This makes it compatible with the ECLIPSE 300 form of the keyword. It is now the same as the COMPDATM keyword in ECLIPSE 100, which is treated as an alias for COMPDATL.

For back compatibility, ECLIPSE 100 still recognizes and reads the old form of COMPDATL if this is entered. The form of the keyword (as in 2000A and after) is distinguished by item 6 being an integer (it must not be defaulted), or alternatively item 2 being a character string which is not a pure integer, or item 7 being a character string beginning with a letter. If any of these conditions is true ECLIPSE 100 recognizes the keyword as the new form, otherwise it reads it as an old form of the keyword.

The keyword is followed by any number of records, described below, each record terminated by a slash (/). A record can be terminated early using a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Local grid name

   The local grids that the well is completed in must belong to the same amalgamation (see keyword AMALGAM).

3. I - location of the connecting grid block(s) within the local grid

   This item can be set to 0 or defaulted if the local grid named in item 2 is the same as the well’s local grid entered in WELSPECL; the I - location of the well head (entered in keyword WELSPECL) will then be used.

4. J - location of the connecting grid block(s) within the local grid

   This item can be set to 0 or defaulted if the local grid named in item 2 is the same as the well’s local grid entered in WELSPECL; the J - location of the well head (entered in keyword WELSPECL) will then be used.

5. K - location of upper connecting block in this set of data within the local grid

   This item cannot be defaulted.

6. K - location of lower connecting block in this set of data within the local grid

   This item cannot be defaulted.
The following data then applies to each layer (or connection) with K - locations between the upper and lower values given above.

7. Open/shut flag of connection

**OPEN**  Connection open to flow

**SHUT**  Connection closed off

**ECLIPSE 100 only**  
**AUTO**  Connection initially closed, but is opened automatically when another connection in the well is closed during an automatic workover. A connection on **AUTO** is opened each time the well is worked over, in the order in which the connections are first defined in **COMPDATL**. Connections on **AUTO** is also opened when necessary to maintain group production rate targets if the **REPERF** action is selected in keyword **PRORDER**.

**ECLIPSE 300 Thermal**  
**HEAT**  Connection is defined to model heat transfer to the formation when using the multisegment well model. Heat connections cannot be opened to fluid flow during the simulation.

**ECLIPSE 300 only**  
In pre-2012.1 releases, any input other than **SHUT** would result in an **OPEN** connection. This behavior has now been changed so that only **OPEN**, **SHUT** or **HEAT** may be input.

**DEFAULT:**  **OPEN**

8. Saturation table number for connection relative permeabilities

If a default, zero or negative value is entered, the relative permeabilities are calculated using the same saturation table as the grid block containing the connection, selected with keyword **SATNUM**.

In runs using the hysteresis option (keyword **SATOPTS** in the **RUNSPEC** section), the saturation table number entered here will apply to both imbibition and drainage processes. If a different table number is required for imbibition, it should be set using **COMPIMB** after the **COMPDATL** keyword. If a default, zero or negative value is entered, the respective tables for the grid block containing the connection (defined using **SATNUM** for the drainage table and **IMBNUM** for the imbibition table) will be used.

**ECLIPSE 100**  
In runs using the vertical equilibrium option (keyword **VE** in the **RUNSPEC** section) the end-points of the saturation table are used to calculate the contact depths and relative permeabilities at the well connection.

**ECLIPSE 300**  
If the **ODD3P** relative permeability and capillary pressure hysteresis model has been selected and a non-default saturation table number is specified, this will be ignored.

9. Transmissibility factor for the connection

If defaulted or set to zero, the connection transmissibility factor are calculated using the remaining items of data in this record. See "Well inflow performance" in the **ECLIPSE Technical Description** for an account of the methods used in Cartesian and radial geometries. The well bore diameter must be set in item 10.

**UNITS:**  cP.rm$^3$/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm$^3$/day/atm (PVT-M)

Note the following two points:

- If the well is situated on an edge (or a corner) of a Cartesian grid, for example in a sector model or symmetry element, use keyword **WPIMULTL** after **COMPDATL** to scale the resulting connection factors by 0.5 (or 0.25).
• A well connection in an outermost block of a radial grid (I = NDIVIX, which is specified in RUNSPEC keyword DIMENS) is assumed to occupy the entire outer boundary of the block for the purpose of applying boundary conditions. If you wish to model a real well in an outer block of a radial grid, the calculated connection factor is much too large. In this case, you should either supply an appropriate connection factor value yourself, or add an extra ring of inactive cells so that the well is no longer in an outer block.

If a value > 0.0 is entered, the connection transmissibility factor is set to this value. The value is also used to calculate either the connection's effective Kh or the denominator term \( \ln(r_o/r_w) + S \) according to the following rules:

• If the Kh value in item 11 is defaulted or set negative, and the well bore diameter set in item 10, the Kh value is calculated from the connection transmissibility factor and the denominator term. In this case, however, the Kh is only used for output (if the mnemonic WELSPECS is entered in RPTSCHED). The denominator term is calculated from the grid block dimensions, as described in the "Well inflow performance" in the ECLIPSE Technical Description.

• If the Kh value is set > 0.0, the denominator term will be calculated from the connection transmissibility factor and the given Kh value, instead of the well bore radius and the grid block dimensions.

• If the Kh value is set = 0.0, it is replaced by the effective Kh of the grid block, then used to calculate the denominator term in the same way as if it had been set > 0.0.

• The recalculation of Kh or the denominator term may be turned off by setting item 45 of the OPTIONS keyword > 0. The denominator term is then always be calculated from the grid block dimensions and the well bore radius. This option can be used to achieve back-compatibility with pre-95A datasets, if required.

The denominator term is used when calculating the well's productivity/injectivity index and when calculating the effects of a D-factor (see "Well inflow performance" in the ECLIPSE Technical Description). Thus, when the connection factor is set to a positive value, the results of these calculations will depend on the setting of the Kh value in item 11.

**Note:** If a positive value is entered for the connection transmissibility factor, this should include the effect of any skin entered in item 12, but exclude the effect of any D-factor. The effect of a D-factor is modeled separately by adjusting the mobility (see "The flow-dependent skin factor" in the ECLIPSE Technical Description).

10. Well bore diameter at the connection

   This quantity is required for:

   • Calculating the connection transmissibility factor
     (If a zero or negative value was supplied for item 9)
   • Calculating the Well’s Productivity/Injectivity Index
     (If a positive value was supplied for the drainage radius in item 8 of keyword WELSPECL)
   • Calculating the effects of a D-factor
   • Calculating the equivalent global well’s connection transmissibility factor.

   **UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

   **DEFAULT:** 1 ft = 0.3048 m
11. Effective Kh (permeability x thickness) value of the connection

If a default, zero or negative value is entered, the Kh value will be calculated from the grid block data.

If a positive value for the connection transmissibility factor is entered in item 9, the calculation of the denominator term depends on whether the Kh value is set or defaulted. See item 9.

This quantity is required for calculating the connection transmissibility factor (if a default or zero value was supplied for item 9). It is ignored if not required.

**UNITS:** mD.m (METRIC), mD.ft (FIELD), mD.cm (LAB), mD.m (PVT-M)

**DEFAULT:** Negative

12. Skin factor

This quantity is used for the same purposes as listed for item 10. It will be ignored if not required.

Large negative skin factors can cause problems if they increase the effective wellbore radius to approach the pressure equivalent radius of the grid block. Very high values of the connection transmissibility factor may result, which could cause convergence problems. In these circumstances, it may be preferable to change the grid block data near the well to model the physical cause of the negative skin (for example enhance the permeability out to a certain distance from the well). If the effective wellbore radius is increased beyond the pressure equivalent radius, the connection factor becomes negative and an error message is issued.

**ECLIPSE 100**

A warning message is issued if the effective wellbore radius increases beyond half the pressure equivalent radius, or extends outside the grid block.

**DEFAULT:** 0.0

13. D-factor, for handling the effects of non-Darcy flow of free gas

A D-factor measured for the well should preferably be entered with keyword **WDFAC**, and this item should be left defaulted. But it may be entered here instead, for back-compatibility with old ECLIPSE 100 datasets. ECLIPSE converts the well D-factor into D-factors for the connections. See the **WDFAC** keyword for information on how the conversion is performed. Note that the well D-factor must reflect the current number of open connections at the time it is specified.

If you wish to enter the connection’s D-factor directly, you can enter it here with a negative sign. A negative sign tells ECLIPSE to regard it as the connection’s D-factor rather than the well’s D-factor. ECLIPSE simply reverses its sign without applying any conversion.

**UNITS:** day/sm³ (METRIC), day/Mscf (FIELD), hr/scc (LAB), day/sm³ (PVT-M)

**DEFAULT:** “Keep the current value of the connection’s D-factor”

14. Direction in which the well penetrates the grid block

- **X** the well penetrates the grid block in the X direction
- **Y** the well penetrates the grid block in the Y direction
- **Z** the well penetrates the grid block in the Z direction

**ECLIPSE 300 only**

- **FX** the well is completed in a fracture in the X direction

**ECLIPSE 300 only**

- **FY** the well is completed in a fracture in the Y direction
In radial grids, only penetration in the Z direction is allowed.

**ECLIPSE 300**

For Cartesian grids in ECLIPSE 300, if a direction other than Z is specified, the `HWELLS` keyword must be included in the RUNSPEC section. If FX or FY is specified, the keyword `FWELLS` must be included in the RUNSPEC section.

This item is used in calculating the transmissibility factor of the connection if it was not supplied in item 9. In Cartesian grids the Peaceman formula is used, taking the components of permeability and grid block dimensions appropriate to the direction of penetration (see "Well inflow performance" in the ECLIPSE Technical Description). Note that the formula assumes that the well completely penetrates a rectangular block through its center in a direction perpendicular to two of its faces.

This item also influences the way ECLIPSE orders the connections in the well (except for friction and multisegment wells) if the default TRACK method is used. See keyword `COMPORD` for further details.

15. Pressure equivalent radius, \( r_o \)

If a value > 0.0 is entered, \( r_o \) is set to this value instead of being calculated from the grid block dimensions according to Peaceman’s formula (see "Well inflow performance" in the ECLIPSE Technical Description). Note that the pressure equivalent radius is not used for connections in the innermost or outermost cells of a radial grid or, in ECLIPSE 300, for wells completed in fractured cells.

**UNITs:** m (**METRIC**), ft (**FIELD**), cm (**LAB**), m (**PVT-M**)

**DEFAULT:** Calculate \( r_o \) from Peaceman’s formula

**Notes**

- The calculated values of the connection transmissibility factor and \( Kh \) (if defaulted) can be examined by requesting a printout of the SCHEDULE section keyword data (argument WELSPECS in keyword `RPTSCHED`). The connection transmissibility factor can also be written to the SUMMARY file with the SUMMARY section keyword `LCTFAC`.

- If you wish to prevent inter-block flow (for example due to shale breaks or faults), use keywords `MULTZ` and `MULTX` for example to set a zero transmissibility multiplier. Do not set the rock permeability to zero instead, as it is needed to calculate the connection transmissibility factor.

- There is a choice of methods for determining the ordering of the connections along the wellbore. See keyword `COMPORD` for further details.

**Example**

```plaintext
COMPDATL
  PROD1 CARF1 6 4 1 3 OPEN 0 0.0 0.333 500 3.5 1E-3 Z /
  PROD2 CARF2 3 5 3 3 OPEN 3 0.0 0.333 /
  PROD3 CARF3 3 5 2 2 OPEN 0 1* 0.333 3* X /
  PROD3 CARF3 4 5 2 3 OPEN 0 1* 0.333 3* X /
```

**Keywords**

478
COMPDATM  Completion data for wells in amalgamated local refined grids

COMPDATM is an alias for COMPDATL.
COMPFLSH  Flash transformation ratios for well connections

COMPFLSH may be used to specify transformation ratios, to transform well oil and gas production flows from differential liberation conditions to flash liberation conditions. For flexibility, the transformation ratios can be set independently for different well connections (ECLIPSE 100 only). The well connections must already have been defined with the keyword COMPDAT.

COMPFLSH is followed by any number of records, described below, each terminated by a slash (/). The set of records must end with a blank record, containing only a slash (/).

Each record can contain some or all of the following items:

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. I - location of connecting grid block(s)
   This item is not used in ECLIPSE 300 and must be defaulted.
   DEFAULT: 0 (Allows any I-value)

3. J - location of connecting grid block(s)
   This item is not used in ECLIPSE 300 and must be defaulted.
   DEFAULT: 0 (Allows any J-value)

4. K - location of upper connecting block in this set of data
   This item is not used in ECLIPSE 300 and must be defaulted.
   DEFAULT: 0 (Corresponds to top connection of well)

5. K - location of lower connecting block in this set of data
   This item is not used in ECLIPSE 300 and must be defaulted.
   DEFAULT: 0 (Corresponds to bottom connection of well)

The following flash transformation ratios are applied to the currently defined connections in the well which match the location indices specified in items 2 - 5. But if a location index is defaulted to zero, it plays no part in selecting which connections use these ratios. Thus if the I and J location indices are defaulted to zero, the transformation ratios are applied to all connections in the well that are located between layers K1 and K2 specified in items 4 and 5. If all four location indices are defaulted, as required in ECLIPSE 300, the transformation ratios are applied to all the connections in the well.

6. Transformation ratio ($F_1$) for oil formation volume factor at the bubble point, where

$$F_1 = \frac{B_{ob}(\text{flash}) - 1}{B_{ob}(\text{diff}) - 1}$$

Eq. 3.18

DEFAULT: 1.0

7. Transformation ratio ($F_2$) for solution gas - oil ratio, where
Flash PVT table number.

If a number > 0 is entered, the transformation ratios entered in items 6 and 7 will be ignored, and instead the ratios are calculated at the beginning of each timestep from the respective equations using the properties of bubble point oil. $B_{ob}(\text{diff})$ and $R_s(\text{diff})$ are obtained from the PVT table of the grid block containing the connection (see keyword \texttt{PVTNUM}) at its bubble point pressure. $B_{ob}(\text{flash})$ and $R_s(\text{flash})$ are the properties of saturated oil, at the same bubble point pressure, obtained from the nominated flash PVT table.

A flash PVT table cannot be used in API tracking runs.

DEFAULT: 0

Note: Only the saturated oil properties in this table (keyword \texttt{PVCO} or \texttt{PVTO}) will be used in the calculation. The other PVT property keywords (for example \texttt{PVTW}, \texttt{PVTG}, \texttt{DENSITY}, \texttt{ROCK}) must also have a table corresponding to this table number, but these tables will not be used. They could be defaulted, or contain copies of the table number \texttt{PVTNUM}.

Example

```
COMPFLUSH
  -- well  I  J  K1  K2  F1  F2  flash
  -- name  table
  PROD1  4*  1.2  1.15 / all connections use these ratios
  PROD2  2*  1  3  1*  1*  2 / only layers 1-3 use this table
/
```

Treatment of flash transformation ratios in ECLIPSE

Within an oil reservoir, the release of dissolved gas from the oil as the pressure decreases resembles a differential liberation process, in which the gas is removed from the oil as it is released. By contrast, the continuing release of dissolved gas from the oil in its passage up the wellbore and through the separator resembles a flash liberation process, in which released gas remains in contact with the oil. Often it is possible to account for these different processes by devising a suitable composite PVT table to use in both the well and the reservoir. But this cannot be done if the bubble point pressure varies significantly within the reservoir.

The flash transformation facility provides an alternative solution to this problem. The oil PVT tables for the reservoir are constructed to model a differential liberation and a transformation is applied to the well rates to account for the effects of flashing the fluid from local reservoir conditions. The transformation parameters are specified in the form of two ratios:

$$F_1 = \frac{B_{ob}(\text{flash}) - 1}{B_{ob}(\text{diff}) - 1} \quad F_2 = \frac{R_s(\text{flash})}{R_s(\text{diff})}$$

where

$B_{ob}$ is the formation volume factor of oil at bubble point pressure, and
\( R_s \) is the solution gas oil ratio.

Using these ratios will ensure that \( B_{ob} \) (flash) approaches 1.0 and \( R_s \) (flash) approaches zero as their differential counterparts do when the pressure approaches stock tank conditions. At pressures above the bubble point, the ratio of oil formation volume factors is maintained the same as that for bubble point oil:

\[
\frac{B_o \text{ (flash)}}{B_o \text{ (diff)}} = \frac{B_{ob} \text{ (flash)}}{B_{ob} \text{ (diff)}} = F_1 + \frac{1-F_1}{B_{ob} \text{ (diff)}}
\]

Eq. 3.21

This leaves the compressibility of undersaturated oil unaltered by the transformation.

### Transforming the oil and gas flows

The oil and gas flow rates entering the well through the connections are calculated as if \( B_{o} \) (flash) and \( R_s \) (flash) were used to determine the phase mobilities. The flows within the reservoir, and the flows leaving the reservoir through the well connections, are calculated using \( B_{o} \) (diff) and \( R_s \) (diff), as given in the oil PVT table. ECLIPSE does this by transforming the oil and gas flows that have entered the well \((Q_o, Q_g)\) to \((Q_o^*, Q_g^*)\), where

\[
Q_o^* = T_{\text{oil-oil}}Q_o
\]

\[
Q_g^* = T_{\text{gas-oil}}Q_o + Q_g
\]

with the transformation coefficients

\[
T_{\text{oil-oil}} = \frac{1}{F_1 + \left( \frac{1-F_1}{B_{ob}\text{ (diff)}} \right)}
\]

Eq. 3.22

\[
T_{\text{gas-oil}} = (T_{\text{oil-oil}}F_2 - 1)R_s \text{ (diff)}
\]

Eq. 3.23

The transformation coefficients for each well connection are calculated at the start of each timestep from the values of \( B_{ob} \) and \( R_s \) in the connecting grid blocks. They are then held constant for the duration of the timestep. The use of explicit \( B_{ob} \) and \( R_s \) values in the transformation coefficients is not expected to cause serious oscillations; however, if these do occur, they may be damped out by specifying a time-average weighting for these quantities with item 42 in the options keyword (ECLIPSE 100 only).

### Reporting

The flow transformations are applied inside the well bore. Thus the well flows reported in the Print and Summary files have been transformed to flash conditions, while the connection flows reported are the original differential values. The group and field flows, being the sum of the well flows, are also reported at flash conditions. All well, group and field rate constraints are applied to the flows transformed to flash conditions.

Reporting the connection flows at their original differential values has a number of advantages. They, and their derived quantities such as layer and region production, can be directly compared with the fluids in place, which are reported at differential liberation conditions. Also, in the absence of free gas flow, the reported connection GOR is equal to the \( R_s \) of the oil in the grid block. Note, however, that the reported well flow is no longer the exact sum of the reported connection flows, the difference being the amount of fluid lost or gained by the transformation to flash conditions.
In the fluid-in-place reports, the fluids in place and all flows are reported at differential liberation conditions.

**Comparison with SEPVALS**

The **SEPVALS** keyword, which changes the separator conditions, provides an alternative means of transforming the oil and gas flows. The two options are incompatible, and should not be used in the same run.

There are a number of differences between the **SEPVALS** and **COMPFLSH** facilities:

- **SEPVALS** applies the same transformation to the well flow and all the well’s connection flows. **COMPFLSH** can apply different transformations to individual connections (useful when a well is completed in two regions with different types of oil).

- **SEPVALS** transforms the flows by applying a constant ratio to $B_0$ and a constant increment to $R_s$. If the bubble point pressure falls significantly during the simulation, the transformation will become less accurate, because the transformed values of $B_0$ and $R_s$ do not approach 1.0 and 0.0 respectively as the pressure reduces to surface conditions. The **COMPFLSH** transformation, on the other hand, applies constant ratios to $B_{ob} - 1$ and $R_s$, so their transformed values will approach the correct values as the pressure reduces to surface conditions.

- With **SEPVALS**, both the well and connection flows are reported at the transformed rates. With **COMPFLSH**, the connection flows are reported at the untransformed rates.

- With **SEPVALS**, the VFP tables are interpolated using the untransformed well rates (at original separator conditions), as it is assumed that the tables were prepared at the original separator conditions. With **COMPFLSH** the tables are interpolated using the well rates transformed to flash conditions, as it is assumed that they were prepared using flash liberation PVT properties.

**Restrictions and limitations**

- The flash transformation facility should only be used in black oil runs having both live oil (containing dissolved gas) and dry gas as active phases.

- The facility may not be used in gas condensate runs, nor in conjunction with the miscible flood and solvent options.

- A flash PVT table (see item 8) cannot be used in API tracking runs, but it is still possible to specify transformation ratios in items 6 and 7.

- It is strongly recommended that the flash transformation facility is not used on connections of multisegment wells in ECLIPSE 100 (this option is not permitted in ECLIPSE 300).

In both the conventional well and the multisegment well models the inflowing fluid is transformed to flash conditions when it enters the wellbore through the connections to the reservoir grid. To calculate the fluid properties (for example, for the hydrostatic head) a back transformation is applied based on the overall inflow; this is consistent with the treatment of the well as a “mixing tank” and is thus appropriate for the conventional well model. With a multisegment well, this back transformation (based on the overall inflow) is still applied, even though the local fluid properties may vary with position in the well. Thus the local fluid properties are not as accurately represented as they should be for a discretized wellbore model such as a multisegment well. These fluid properties influence the hydrostatic and frictional pressure gradients, the out flowing mixture properties when crossflow occurs, and slip (when using the drift-flux model).
COMPIMB is used to set the imbibition saturation table number for well connections, in runs using the Hysteresis option (keyword SATOPTS in the RUNSPEC section). The table number specified in item 7 of keyword COMPDAT applies to both the imbibition and drainage processes. If a different table number is required for imbibition, it should be set using COMPIMB after the COMPDAT keyword.

If the ODD3P relative permeability and capillary pressure hysteresis model has been specified this keyword will be ignored.

COMPIMB is followed by any number of records, described below, each record terminated by a slash (/). A record may be terminated early at any item; the remaining items will assume their default values. The set of records must end with a blank record, containing only a slash (/).

Each record can contain some or all of the following items:

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. I - location of connecting grid block(s)

   DEFAULT: 0 (allows any I-value)

3. J - location of connecting grid block(s)

   DEFAULT: 0 (allows any J-value)

4. K - location of upper connecting block in this set of data

   DEFAULT: 0 (corresponds to top connection of well)

5. K - location of lower connecting block in this set of data

   DEFAULT: 0 (corresponds bottom top connection of well)

   The table number in item 6 is applied to all the connections in the well which match the location indices specified in items 2 - 5. But if a location index is defaulted to zero, it plays no part in selecting the connections to which the table number applies. Thus if the I and J location indices are defaulted to zero, the table number is applied to all connections in the well that are located between layers K1 and K2 specified in items 4 and 5. If all four location indices are defaulted, the table number is applied to all the connections in the well.

6. Imbibition saturation table number for the connection(s)

   If the table number is zero, the imbibition table for the grid block containing the connection (set using keyword IMBNUM) will be used.

   DEFAULT: 0

**Example**

The first line of data sets the imbibition table number to 3 for all the connections belonging to well PROD1. The second line of data sets the imbibition table number to 2 for the layer-3 connection of well PROD2.

| COMPIMB | PROD1 | 4* | 3 | / |
PROD2  2*  3   3  2  /

In well connections where fluid is being injected into the formation, the mobility of the injected phase is made to vary as the total fluid mobility in the grid block (see "Injecting connections" in the ECLIPSE Technical Description),

\[
M(p, j) = \frac{\left[ k_r(o, j) \mu(o, j) + k_r(w, j) \mu(w, j) + k_r(g, j) \mu(g, j) \right]}{B(p, j)}
\]  
Eq. 3.24

If gas or water is injected into a grid block initially containing oil, this relationship will cause the well’s injectivity to vary until the grid block is completely flooded out with the injected phase. In reality however, most of the pressure drop occurs over a region close to the well, and when this region has flooded out the injectivity stays approximately constant. If the grid block size is much larger than this region, the calculated injectivity is incorrect until the whole grid block has flooded out.

The COMPINJK keyword provides an alternative approach, which is more accurate in cases where:

- The well injects fluid having a mobility significantly different to that of the fluid initially in the grid block, and
- The grid blocks containing the well connections are large, and
- The well is not subject to crossflow.

The relative permeability of the injected phase is given a constant value \( k_r(p,*) \), representing its value in the region that has been flooded out by the injected phase. The relative permeabilities of the other phases are set to zero. The relationship for the injected phase mobility thus becomes

\[
M(p, j) = \frac{k_r(p, *)}{B(p, j) \mu(p, j)}
\]  
Eq. 3.25

COMPINJK is followed by any number of records, each containing some or all of the following items of data and terminated by a slash (/). The set of records must end with a blank record, containing only a slash.

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. I - location of connecting grid block(s)
   DEFAULT: 0 (allows any I-value)

3. J - location of connecting grid block(s)
   DEFAULT: 0 (allows any J-value)

4. K - location of upper connecting block in this set of data
   DEFAULT: 0 (corresponds to top connection of well)

5. K - location of lower connecting block in this set of data
   DEFAULT: 0 (corresponds to bottom connection of well)

6. Relative permeability of the injected phase
The relative permeability in item 6 is applied to all the connections in the well which match the location indices specified in items 2 - 5. But if a location index is defaulted to zero, it plays no part in selecting which connections use the relative permeability value. Thus if the I and J location indices are defaulted to zero, the relative permeability value is applied to all connections in the well that are located between layers K1 and K2 specified in items 4 and 5. If all four location indices are defaulted, the relative permeability value is applied to all the connections currently defined in the well.

A positive value is applied as the relative permeability of the injected phase.

A default or negative value causes the relative permeability of the injected phase to be set to a suitable value in the saturation table used by the well connection, depending on the phase of the injector:

\[ k_{rw}(S_w = 1 - S_{owcr} - S_{gco}) \text{ for water injectors} \]
\[ k_{rg}(S_g = 1 - S_{oger} - S_{wo}) \text{ for gas injectors} \]
\[ k_{ro}(S_w = S_{wco}, S_g = S_{gco}) \text{ for oil injectors.} \]

If a default or negative value is supplied, the well must already have been declared an injector with keyword \texttt{WCONINJE} in order to know what phase it injects.

A zero value will cause the injected phase mobility to vary as the total fluid mobility in the grid block, resorting to the standard relationship applied in the absence of keyword \texttt{COMPINJK}.

\textit{Note:} An injection well with a constant user-defined value of the injected phase relative permeability must not be allowed to crossflow, as this may result in multiphase injection. If a well has its crossflow flag set to \texttt{YES} (item 10 in keyword \texttt{WELSPECS}), its flag will be reset to \texttt{NO} and a warning message is printed.

\textbf{Example}

This example causes all connections in well \texttt{INJ1} to have a constant injected phase relative permeability set to the appropriate saturation table end point. The connection in layer 3 of well \texttt{INJ2} has an injected phase relative permeability of 1.0.

\begin{verbatim}
COMPINJK
INJ1              /
INJ2  2* 3 3 1.0  /
/ 
\end{verbatim}
Relative permeabilities for injector

**COMPKRI** is used to set the phase relative permeabilities for an injection well completion. The total mobility is

\[
\frac{k_{rw}}{\mu_w} + \frac{k_{ro}}{\mu_o} + \frac{k_{rg}}{\mu_g}
\]

Eq. 3.26

Setting the phase relative permeabilities will fix the relative permeabilities in the total mobility while allowing the viscosities to vary implicitly. Injection will then be the product of a mass or energy density times the total mobility times the connection factor times the drawdown (see "Injecting connections" in the *ECLIPSE Technical Description*).

The keyword is followed by any number of records, each containing the items of data described below, and each terminated by a / character.

The records can be terminated early using the / character, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. **Well name, well name template, well list or well list template**
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the *ECLIPSE Technical Description* for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword **WLIST**.

2. **I - location of connecting grid block(s)**
   DEFAULT: 0 (allows any I-value)

3. **J - location of connecting grid block(s)**
   DEFAULT: 0 (allows any J-value)

4. **K - location of upper connecting block in this set of data**
   DEFAULT: 0 (corresponds to top connection of well)

5. **K - location of lower connecting block in this set of data**
   DEFAULT: 0 (corresponds to bottom connection of well)

6. **Kro**, the oil relative permeability.
   A negative value will leave a previously defined value unaltered.
   DEFAULT: 0.5

7. **Krg**, the gas relative permeability.
   A negative value will leave a previously defined value unaltered.
   DEFAULT: 0.5

8. **Krw**, the water relative permeability.
   A negative value will leave a previously defined value unaltered.
   DEFAULT: 0.5
9. Only use the relative permeability of the injected phase in the injection mobility calculation

   YES  The relative permeabilities of the phases not injected by the well will be set to zero in this calculation.

   NO   The values as entered in items 6, 7 and 8 will be used in this calculation.

   DEFAULT: NO

   To retrospectively set this item to YES for previously defined COMPKRI keywords, default the connection items appropriately and set the relative permeability items 6 to 8 negative to ensure your values are not overwritten with default values.

   **Note:** This option can introduce non-linearities into the model. The problem is that if a phase in not present in a cell then it wouldn’t contribute to the injection mobility (note that the phase viscosity is not known). However, if a minute amount of the phase is present then the injection mobility will increase to reflect the rel-perm input in COMPKRI. This non-linearity causes the simulator significant convergence problems. Hence it is recommended that only the phase relative permeability of either the injected phase and/or a phase likely to exist in the reservoir should be specified. So, for example, items 6 and 7 could be set to zero for water injectors. Alternatively, setting item 9 to YES would ensure that while the well injects water, items 6 and 7 are effectively zero, and while it injects gas, items 6 and 8 are effectively zero.

   **Example**

   ```
   COMPKRI
   \[ \text{"WINJ" } 4* \ 0.0 \ 0.0 \ 0.8 / \]
   \[ \text{"CYCLIC" } 4* \ 0.4 \ 0.0 \ 0.2 / \]
   ```

   **Keywords**

   489
Relative permeabilities for injector completions in local refined grids

COMPKRIL is used to set the phase relative permeabilities for an injection well completion in a local refined grid (LGR) cell. The keyword data is similar to that for COMPKRI, except that there is an additional item at position 2 which gives the name of the LGR in which the completion is located.

The total mobility is

\[
\frac{k_{rw}}{\mu_w} + \frac{k_{ro}}{\mu_o} + \frac{k_{rg}}{\mu_g}
\]

Eq. 3.27

Setting the phase relative permeabilities will fix the relative permeabilities in the total mobility while allowing the viscosities to vary implicitly. Injection will then be the product of a mass or energy density times the total mobility times the connection factor times the drawdown (see "Injecting connections" in the ECLIPSE Technical Description).

The keyword is followed by any number of records, each containing the items of data described below, and each terminated by a / character.

The records can be terminated early using the / character, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Local grid name

3. I - location of connecting grid block(s)
   DEFAULT: 0 (allows any I-value)

4. J - location of connecting grid block(s)
   DEFAULT: 0 (allows any J-value)

5. K - location of upper connecting block in this set of data
   DEFAULT: 0 (corresponds to top connection of well)

6. K - location of lower connecting block in this set of data
   DEFAULT: 0 (corresponds to bottom connection of well)

7. Kro, the oil relative permeability.
   A negative value will leave a previously defined value unaltered.
   DEFAULT: 0.5

8. Krg, the gas relative permeability.
   A negative value will leave a previously defined value unaltered.
   DEFAULT: 0.5
9. Krw, the water relative permeability.
   A negative value will leave a previously defined value unaltered.
   DEFAULT: 0.5

10. Only use the relative permeability of the injected phase in the injection mobility calculation
    YES  The relative permeabilities of the phases not injected by the well will be set to zero in this calculation.
    NO   The values as entered in items 7, 8 and 9 will be used in this calculation.

    DEFAULT: NO

To retrospectively set this item to YES for previously defined COMPKRI keywords, default the connection items appropriately and set the relative permeability items 7 to 9 negative to ensure your values are not overwritten with default values.

Note: This option can introduce non-linearities into the model. The problem is that if a phase in not present in a cell then it wouldn’t contribute to the injection mobility (note that the phase viscosity is not known). However, if a minute amount of the phase is present then the injection mobility will increase to reflect the relative permeability input in COMPKRI. This non-linearity cause the simulator significant convergence problems. Hence it is recommended that only the phase relative permeability of either the injected phase and/or a phase likely to exist in the reservoir should be specified. So, for example, items 7 and 8 could be set to zero for water injectors. Alternatively, setting item 10 to YES would ensure that while the well injects water, items 7 and 8 are effectively zero, and while it injects gas, items 7 and 9 are effectively zero.

Example

```
COMPKRI
CYCLIC LGR1 4* 0.4 0.2 0.1 /
/
```
COMPLMPL  Lumps connections in local grids for automatic workovers

COMPLMPL is used to lump well connections together into completions for simultaneous closure in automatic workovers, when the connections are located in local grid refinements. The connections must have first been defined with the keyword COMPDATL. COMPLMPL performs the same role as COMPLUMP does, but the name of the local grid containing the connections specified in the record should be entered in item 2 of COMPLMPL.

Wells may be worked over after violating limits set in keywords WECON, CECON, GECON, GCONPROD, GCONPRI and, for ECLIPSE 100, GCONSAL, when for example the worst offending connection will be shut. If the connections have been lumped into completions, then the worst-offending completion will be identified (by summing the production rates of its connections) and all the connections belonging to it will be shut together. The connection economic limits in keyword CECON will also be applied to completions, that is a whole completion will be closed if its total water cut, GOR or WGR exceeds the limit set for its connections.

If a worked-over well has any connections left in the AUTO queue for automatic opening (see keyword COMPDATL. COMPDATM is just an alias for COMPDATL), then the completions are examined in increasing order of their completion number, and the first one found with connections in the AUTO queue will have all its connections opened together.

When a connection is first defined with keyword COMPDATL, it is given a completion number equal to its order of introduction in the well. For example, the third connection defined in the well is put in completion number three. When several connections are defined in one record (items 4 and 5 define the upper and lower layers), the connections are introduced in layer order from top to bottom. So if the keyword COMPLMPL is not used, each connection is in its own separate completion, and the terms completion and connection are equivalent.

**Note:** When the COMPLMPL keyword is used, the named connections will be added to any existing connections in the completion specified. For this reason, it is advisable that the completion numbers of all connections in the well are re-specified at the same time.

**Note:** If a well is completed in several amalgamated local grids, then connections in different local grids may not be lumped together. Only connections within the same local grid may be lumped.

Completion numbers can also be used in the WEOPENL, WPIMULTL and WSCCLENL keywords to refer to a range of connections. Also, they can be used in the SUMMARY section with the keywords beginning with the letter W and ending with the letter L to output completion quantities. Keywords beginning with the letters LC and ending with the letter L output the corresponding connection quantity totaled over all the connections that belong to the same lumped completion located in local grid refinements. Any connection within the lumped completion can be specified to define the completion. For example, WOFRL and LCOFRL give the oil flow rate for the entire completion, though note that, in the latter case, the completion is identified by specifying a connection within the completion and not by specifying the completion number itself. See the SUMMARY section documentation for a full list of completion keywords.

COMPLMPL is followed by any number of records, described below, each record terminated by a slash (/). The set of records must end with a blank record, containing only a slash.

Each record contains the following items

**Keywords**
1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Name of local grid containing the completions to be lumped

   If this item is defaulted or set to a blank string, the local grid name defaults to the well’s local grid named in keyword WELSPECL.

   This item may not be defaulted, however, if the local grid containing the well is amalgamated (using keyword AMALGAM).

   The connection locations in items 3 - 6 refer to their coordinates within this local grid.

3. I - location of connecting grid block(s)

   DEFAULT: 0 (allows any I-value)

4. J - location of connecting grid block(s)

   DEFAULT: 0 (allows any J-value)

5. K - location of upper connecting block in this completion

   DEFAULT: 0 (corresponds to top connection of the well in the local grid specified in item 2)

6. K - location of lower connecting block in this completion

   DEFAULT: 0 (corresponds to bottom connection of the well in the local grid specified in item 2)

7. Completion number of the connections

   This must lie between 1 and NCWMAX (set by item 2 of keyword WELLDIMS in RUNSPEC).

   The set of connections in the well which match the location indices specified in items 3 to 6 will be given the completion number specified in item 7. All connections having the same completion number are treated as belonging to the same completion, and will be closed together in an automatic workover.

   If a location index is defaulted to zero, it plays no part in selecting the set of connections. Thus if the I and J location indices are defaulted to zero, all connections in the well that are located between layers K1 and K2 specified in items 5 and 6 will be given the completion number specified in item 7. If all four location indices are defaulted, all the well’s connections will be given this completion number.

Note: It is still possible to use keyword COMPLUMP for wells completed in a single, unamalgamated local grid.

Example

```
COMPLMPL
-- Connections in layers 1 and 2 for local grid CARF1
-- are in completion 1 for well PROD1
   PROD1  CARF1  2*  1  2    1 /
-- Connections in layers 3 and 4 for local grid CARF2
-- are in completion 2 for well PROD1
   PROD1  CARF2  2*  3  4    2 /
-- Connections in layer 5 for local grid CARF3
-- are in completion 3 for well PROD2
```

Keywords

493
COMPLUMP Lumps connections for automatic workovers

COMPLUMP is used to lump well connections together into completions for simultaneous closure in automatic workovers. The connections must first have been defined with the keyword COMPDAT.

Wells may be worked over after violating limits set in keywords WECON, CECON, GECON, GCONPROD, GCONPRI and, for ECLIPSE 100, GCONSALE, when for example the worst offending connection will be shut. If the connections have been lumped into completions, then the worst-offending completion will be identified (by summing the production rates of its connections) and all the connections belonging to it will be shut together. The connection economic limits in keyword CECON will also be applied to completions, that is a whole completion will be closed if its total water cut, GOR or WGR exceeds the limit set for its connections.

In ECLIPSE 100, if a worked-over well has any connections left in the AUTO queue for automatic opening (see keyword COMPDAT), then the completions are examined in increasing order of their completion number, and the first one found with connections in the AUTO queue will have all its connections opened together.

When a connection is first defined with keyword COMPDAT, it is given a completion number equal to its order of introduction in the well. For example, the third connection defined in the well is put in completion number three. When several connections are defined in one record (items 4 and 5 define the upper and lower layers), the connections are introduced in layer order from top to bottom. So if the keyword COMPLUMP is not used, each connection is in its own separate completion, and the terms completion and connection are equivalent.

Note: When the COMPLUMP keyword is used, the named connections will be added to any existing connections in the completion specified. For this reason, it is advisable that the completion numbers of all connections in the well are re-specified at the same time.

Completion numbers can also be used in the WELOPEN, WPIMULT and WSCCLEAN keywords to refer to a range of connections. Also, they can be used in the SUMMARY section with the keywords beginning with the letter W and ending with the letter L to output completion quantities. Keywords beginning with the letter C and ending with the letter L output the corresponding connection quantity totaled over all the connections that belong to the same lumped completion. Any connection within the lumped completion can be specified to define the completion. For example, WOFRL and COFRL give the oil flow rate for the entire completion, though note that, in the latter case, the completion is identified by specifying a connection within the completion and not by specifying the completion number itself. See the SUMMARY section documentation for a full list of completion keywords.

COMPLUMP is followed by any number of records, described below, each record terminated by a slash (/).

The set of records must end with a blank record, containing only a slash.

Each record contains the following items:

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. I - location of connecting grid block(s)
   DEFAULT: 0 (allows any I-value)
3. J - location of connecting grid block(s)
   DEFAULT: 0 (allows any J-value)

4. K - location of upper connecting block in this completion
   DEFAULT: 0

5. K - location of lower connecting block in this completion
   DEFAULT: 0

6. Completion number of the connections
   This must lie between 1 and NCWMAX (set by item 2 of keyword WELLDIMS in RUNSPEC).

   The set of connections in the well which match the location indices specified in items 2 to 5 will be
   given the completion number specified in item 6. All connections having the same completion number
   are treated as belonging to the same completion, and will be closed together in an automatic workover
   (see example).

   If a location index is defaulted to zero, it plays no part in selecting the set of connections. Thus if the I
   and J location indices are defaulted to zero, all connections in the well that are located between layers
   K1 and K2 specified in items 4 and 5 will be given the completion number specified in item 6. If all
   four location indices are defaulted, all the well’s connections will be given this completion number.

   **Note:** It is still possible to use keyword **COMPLUMP** for wells completed in a single, non-amalgamated local
   grid.

**Example**

```plaintext
COMPLUMP
-- connections in layers 1 and 2 are lumped with connection 1 in
-- completion 1
   PROD1 2* 1 2 1 /
-- connections in layers 3 and 4 are lumped with /connection 2 in
-- completion 2
   PROD1 2* 3 4 2 /
-- the connection in layer 5 is lumped with connection 3 in completion3
   PROD1 2* 5 5 3 /
```

**Keywords**

496
COMPMBIL  Voidage mobility for injector completions in local refined grids

COMPMBIL is used to set the total mobility for an injection completion in a local refined grid (LGR) cell. The keyword data is similar to that for COMPMOBI, except that there is an additional item at position 2 which gives the name of the LGR in which the completion is located.

The total mobility is

\[ \frac{k_{rw}}{\mu_w} + \frac{k_{ro}}{\mu_o} + \frac{k_{rg}}{\mu_g} \]

Eq. 3.28

Injection will then be the product of a mass or energy density times the total mobility times the connection factor times the drawdown (see "Injecting connections" in the ECLIPSE Technical Description).

The keyword is followed by any number of records, each containing the items of data described below, and each terminated by a / character.

The records can be terminated early using the / character, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Local grid name

3. I - location of connecting grid block(s)

   DEFAULT: 0 (allows any I-value)

4. J - location of connecting grid block(s)

   DEFAULT: 0 (allows any J-value)

5. K - location of upper connecting block in this set of data

   DEFAULT: 0 (corresponds to top connection of well)

6. K - location of lower connecting block in this set of data

   DEFAULT: 0 (corresponds to bottom connection of well)

7. Total voidage mobility as described above.

   UNITS: cP^-1 (METRIC), cP^-1 (FIELD), cP^-1 (LAB), cP^-1 (PVT-M)

   DEFAULT: 10.0 cP^-1
Example

```
COMPMBIL
ICYC5 LGR1 2 2 1 3 10.0 /
```

Keywords

498
**COMPMOBI**  
**Voidage mobility for injector**

COMPMOBI is used to set the total mobility for an injection completion. The total mobility is

\[
\frac{k_{rw}}{\mu_w} + \frac{k_{ro}}{\mu_o} + \frac{k_{rg}}{\mu_g}
\]

Eq. 3.29

Injection will then be the product of a mass or energy density times the total mobility times the connection factor times the drawdown (see "Injecting connections" in the ECLIPSE Technical Description).

The keyword is followed by any number of records, each containing the items of data described below, and each terminated by a / character.

The records can be terminated early using the / character, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword **WLIST**.

2. I - location of connecting grid block(s)
   DEFAULT: 0 (allows any I-value)

3. J - location of connecting grid block(s)
   DEFAULT: 0 (allows any J-value)

4. K - location of upper connecting block in this set of data
   DEFAULT: 0 (corresponds to top connection of well)

5. K - location of lower connecting block in this set of data
   DEFAULT: 0 (corresponds to bottom connection of well)

6. Total voidage mobility as described above.
   UNITS: cP\(^{-1}\) (METRIC), cP\(^{-1}\) (FIELD), cP\(^{-1}\) (LAB), cP\(^{-1}\) (PVT-M)
   DEFAULT: 1.0 cP\(^{-1}\)

**Example**

```
COMPMOBIICYC5 18 6 8 23 10.0 /
```

**Keywords**

499
**COMPOFF**

**Switches off automatic compressors in the network**

The COMPOFF keyword switches off all automatic compressors defined by the GASFCOMP keyword in the Gas Field Operations Model, or the NETCOMPA keyword in the Extended Network Model, except those defined to stay on permanently (PERM in item 9 of NETCOMPA). The compressors will be turned on again automatically as soon as they are needed.

This keyword has no effect on compressors defined with the GNETPUMP keyword.

See "Gas field operations model" and "Network option" in the *ECLIPSE Technical Description* for further details.

The keyword has no associated data.
COMPORD  Defines the ordering of well connections

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Method for ordering the connections in the well

DEPTH  The connections will be ordered according to their vertical depth, from the top downwards. For connections at the same vertical depth, they will be ordered in the sequence they are first declared in COMPDAT or COMPDATL.

INPUT  The connections will be ordered in the sequence that they are first declared in COMPDAT or COMPDATL. If you use this option, you must declare the connections in their correct sequence, starting with the connection nearest the wellhead then working along the wellbore towards the bottom or toe of the well.

TRACK  ECLIPSE will determine the order of the connections by attempting to trace the well track through the grid from the grid blocks in which the connections are located. If this fails, or if all the connections in the well are vertical (according to COMPDAT item 13 or COMPDATL item 14), DEPTH ordering will be used. See the second note below for further explanation.

DEFAULT: TRACK

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Notes

• ECLIPSE uses the connection ordering when outputting RFT/PLT data (see WRFTPLT) and when calculating the wellbore hydrostatic head with the segmented density calculation (WELSPECS item 12 set to SEG). The connection ordering also determines which additional connections are closed in a +CON workover (see for example WECON item 7). In these workovers, all connections below the offending one are also closed. Here, “below” is interpreted as “further from the wellhead according to the connection ordering”. However, if you set item 53 in the OPTIONS to > 0 in ECLIPSE 100 “below” will be interpreted literally and all connections with a greater depth will be closed. For back compatibility in ECLIPSE 300, setting item 2 in this keyword to DEPTH will revert both the +CON workover calculation and the segmented density calculation to their pre-2002A state.

• When the TRACK method is selected, ECLIPSE determines the connection ordering as described below. If all connections in the well penetrate vertically (Z-direction in COMPDAT item 13 or COMPDATL item 14), the connections are ordered according to their depth. But if any connection is declared to be horizontal (X- or Y-direction), ECLIPSE will attempt to order the connections in heel-to-toe fashion. It takes the heel to be the connection whose I,J coordinates are nearest the wellhead I,J (set in WELSPECS items 3 and 4, or WELSPECL items 4 and 5). If more than one connection can be a candidate for the heel, ECLIPSE selects the one with the minimum depth. The rest of the ordering is
determined from the location and orientation of the connections. If ECLIPSE cannot find a unique sensible ordering (for example with a multilateral well), it will issue a warning message and resort to ordering the connections by depth.

- For wells with complex trajectories, and in particular multilateral wells, you are recommended to define these as **multisegment** wells (**WELSEGS**) or, in ECLIPSE 100, **friction** wells (**WFRICTN**). The connection ordering for these wells is determined more precisely by the additional information supplied in their associated keywords. Note that the setting of **COMPORD** item 2 is **ignored** for these types of well; their connection ordering will **always** be determined by a more precise equivalent of the **TRACK** method.

- The **COMPORD** keyword is optional. If it is not entered, all wells will use the **TRACK** method.

**Example**

```plaintext
COMPORD
HWELL1    INPUT /
HWELL2    INPUT /
/
```

Keywords

502
COMPRIV specifies which grid blocks will be allowed to connect to the river bed. This must be entered after the RIVERSYS keyword defining the appropriate river. It must also be defined before the GRDREACH keyword is used for the river.

The keyword is followed by any number of records, described below, each record terminated by a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. River name or river name root
2. I - location of connecting grid block
3. J - location of connecting grid block
4. K - location of connecting grid block

Example

```
COMPRIV
'MYRIVER' 35 1 1 /
'MYRIVER' 35 2 1 /
'MYRIVER' 35 3 1 /
'MYRIVER' 35 4 1 /
'MYRIVER' 35 5 1 /
'MYRIVER' 35 6 1 /
'MYRIVER' 35 7 1 /
'MYRIVER' 35 8 1 /
'MYRIVER' 35 9 1 /
'MYRIVER' 35 10 1 /
'MYRIVER' 35 11 1 /
'MYRIVER' 35 12 1 /
'MYRIVER' 35 13 1 /
/```

Keywords

503
Data for rescaling saturations at well connections

**COMPRP** is used to rescale the fluid saturations at the well connections, to take account of the position of the perforations and the distribution of the phases within large grid blocks. The well connections must already be defined with the keyword **COMPDAT** (or **WELLCOMP**).

**COMPRP** is followed by any number of records, described below, each record terminated by a slash (/). A record may be terminated early at any item; the remaining items will assume their default values. Defaults can also be specified by a null repeat count of the form \( n^* \), where \( n \) is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

*Note:* The **COMPRP** keyword cannot be used with the Hysteresis option active if the imbibition and drainage table numbers are different for any connection (see item 6).

Each record can contain some or all of the following items:

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the **ECLIPSE Technical Description** for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword **WLIST**.

2. **I** - location of connecting grid block(s)
   DEFAULT: 0 (allows any I-value)

3. **J** - location of connecting grid block(s)
   DEFAULT: 0 (allows any J-value)

4. **K** - location of upper connecting block in this set of data
   DEFAULT: 0 (corresponds to top connection of well)

5. **K** - location of lower connecting block in this set of data
   DEFAULT: 0 (corresponds to bottom connection of well)

   The following rescaling quantities are applied to all the connections in the well which match the location indices specified in items 2 - 5. But if a location index is defaulted to zero, it plays no part in selecting which connections use the rescaling data. Thus if the I and J location indices are defaulted to zero, the rescaling data is applied to all connections in the well that are located between layers \( K_1 \) and \( K_2 \) specified in items 4 and 5. If all four location indices are defaulted, the rescaling data is applied to all the connections in the well.

6. Saturation table number for connection relative permeabilities
   If the number is zero or positive, it will override the saturation table number specified in item 7 of keyword **COMPDAT**. If the number is zero, the relative permeabilities will be calculated using the same saturation table as the grid block containing the connection.

   If the number is negative, the saturation table number will not be altered.
In runs using the hysteresis option (keyword SATOPTS in RUNSPEC), the saturation table number entered here will apply to both imbibition and drainage processes. Rescaling is not allowed if the imbibition and drainage table numbers for the connection are different.

DEFAULT: Negative

7. SWMIN Scaled value of the connate water saturation
   The value is ignored in problems not containing water.
   DEFAULT: \( S_{wco} \)

8. SWMAX Scaled value of the maximum water saturation
   The value is ignored in problems not containing water.
   DEFAULT: \( 1 - S_{owcr} \) in three-phase or oil-water problems, or \( 1 - S_{gcr} \) in gas-water problems.

9. SGMIN Scaled value of the critical gas saturation
   The value is ignored in problems not containing both gas and oil.
   DEFAULT: \( S_{gcr} \)

10. SGMAX Scaled value of the maximum gas saturation
    The value is ignored in problems not containing both gas and oil.
    DEFAULT: \( 1 - S_{ogcr} - S_{wco} \)

Example

```
COMPRP
  PROD1 4* 3 .39 .63 .15 .29 / scales all connections in well PROD1
  'P*' 2* 2 2 1* .39 .63 .15 .29 / scales layer-2 connections in all
   wells with names beginning with P.
```

The saturation rescaling equations

\( S_{gcr} = \) Critical gas saturation

\( S_{owcr} = \) Critical oil saturation in water

\( S_{ogcr} = \) Critical oil saturation in gas and connate water

\( S_{wco} = \) Connate water saturation

(See "Saturation functions" in the ECLIPSE Technical Description for a full definition of these quantities)

The relative permeabilities at the well connections are calculated using the scaled values of the water and gas saturations, \( S_{w} \) and \( S_{g} \),

\[
k_{rw} = k_{rw}(SS_{w}) \quad \text{Eq. 3.30}
\]

\[
k_{rg} = k_{rg}(SS_{g}) \quad \text{Eq. 3.31}
\]

\[
k_{ro} = k_{ro}(SS_{w}, SS_{g}) \quad \text{Eq. 3.32}
\]
Water saturation scaling

In three-phase and oil-water problems, the water saturation is scaled as follows:

If

\[ \text{SWMIN} < S_w < \text{SWMAX} \]  \hspace{1cm} \text{Eq. 3.33}

Then

\[ SS_w = (1-f_w)S_{wco} + f_w(1-S_{owcr}) \]

where

\[ f_w = \frac{(S_w - \text{SWMIN})}{(\text{SWMAX} - \text{SWMIN})} \]

If

\[ S_w < \text{SWMIN} \]  \hspace{1cm} \text{Eq. 3.34}

If

\[ S_w > S_{wco} \]

Then \[ SS_w = S_{wco} \]

If \[ S_w < S_{wco} \]

Then \[ SS_w = \frac{(S_w S_{wco})}{\text{MIN}(S_{wco} \text{, SWMIN})} \]

If

\[ S_w > \text{SWMAX} \]  \hspace{1cm} \text{Eq. 3.35}

If \[ S_w < 1-S_{owcr} \]

Then \[ SS_w = 1-S_{owcr} \]

If \[ S_w > 1-S_{owcr} \]

Then \[ 1-SS_w = \frac{(1-S_w)S_{owcr}}{\text{MIN}(S_{owcr} \{1-\text{SWMAX}\})} \]

In gas-water problems, the water saturation is scaled as follows:

If

\[ \text{SWMIN} < S_w < \text{SWMAX} \]  \hspace{1cm} \text{Eq. 3.36}

Then \[ SS_w = (1-f_w)S_{wco} + f_w(1-S_{gcr}) \]

Where \[ f_w = \frac{S_w - \text{SWMIN}}{\text{SWMAX} - \text{SWMIN}} \]

If

\[ S_w < \text{SWMIN} \]  \hspace{1cm} \text{Eq. 3.37}
If $S_w > S_{wco}$
Then $SS_w = S_{wco}$
If $S_w < S_{wco}$
Then $SS_w = \frac{S_w S_{wco}}{\text{MIN}(S_{wco}, SWMIN)}$

If $S_w > SWMAX$  \hspace{1cm} \text{Eq. 3.38}
If $S_w < 1-S_{gcr}$
Then $SS_w = 1-S_{gcr}$
If $S_w > 1-S_{gcr}$
Then $1-SS_w = \frac{(1-S_w)S_{gcr}}{\text{MIN}(S_{gcr}; 1-SWMAX)}$

**Gas saturation scaling**

In three-phase and oil-gas problems, the gas saturation is scaled as follows:

If $SGMIN < S_g < SGMAX$  \hspace{1cm} \text{Eq. 3.39}
Then $SS_g = (1-f_g)S_{gcr} + f_g(1-S_{ogcr} - S_{wco})$

Where $f_g = \frac{(S_g - SGMIN)}{SGMAX-SGMIN}$

If $S_g < SGMIN$  \hspace{1cm} \text{Eq. 3.40}
If $S_g > S_{gcr}$,
Then $SS_g = S_{gcr}$
If $S_g < S_{gcr}$,
Then $SS_g = \frac{S_g S_{gcr}}{\text{MIN}(S_{gcr}, \text{SGMIN})}$

If $S_g > SGMAX$  \hspace{1cm} \text{Eq. 3.41}
If $S_g < 1-S_{ogcr} - S_{wco}$,
Then $SS_g = 1-S_{ogcr} - S_{wco}$
If $S_g > 1-S_{ogcr} - S_{wco}$,
Then \( 1 - SS_g = \frac{(1-S_g)(S_{ogcr} + S_{wco})}{\min\{S_{ogcr} + S_{wco}, 1-S_{gmax}\}} \)

In gas-water problems, the scaled gas saturation is given by

\[ SS_g = 1 - SS_w \]  

**Notes**

- If the default values are used for the items SWMIN, SWMAX, SGMIN and SGMAX, then the saturations are unchanged by the COMPRP keyword.

- If the end point scaling option for grid blocks is being used (keyword ENDScale in RUNSPEC), the end-points used for calculating the default values will be chosen according to the following rules:
  
  - If the saturation table number for the connection (Item 6) is the same as that used for the grid block, the scaled end-points for the grid block will be used.
  
  - If the connection uses a different table from the grid block, the end-points will be unscaled, as specified in the original saturation table data.

  Non-defaulted values specified with COMPRP will override the end-points set for the grid block.

  However, the COMPRP saturation scaling treatment differs from that of the End Point Scaling option. Even when the COMPRP end points are defaulted the results will not be the same, particularly if End Point Scaling scales the critical saturation differently from the connate saturation (COMPRP scales only on the connate water saturation at the lower end). In general, care should be taken when using COMPRP in conjunction with End Point Scaling, and the user must be aware that defaulting the end points will not necessarily reproduce the same results as in the absence of COMPRP.

- In three-phase problems, the gas and water scaling factors must be consistent with each other. If they are not, the scaled oil saturation may become unrealistic. For example, \( SS_w + SS_g \) could exceed 1.0, giving a negative scaled oil saturation. Or \( SS_w + SS_g \) may be less than 1.0 when \( S_o = 0.0 \), which could produce a non-zero oil mobility at the well when there is no oil in the grid block. A warning message will be printed if either situation arises.

**Representing the effects of partial penetration**

Scaled end point saturations can be used to represent the effects of partial penetration in grid blocks where the oil, water and gas phases are assumed to be segregated into zones in vertical equilibrium.

Consider a grid block that originally contained only oil but now has water encroaching from the bottom. Above the water-oil contact the water saturation is \( S_w = S_{wco} \), while below the contact it is \( S_w = 1 - S_{oover} \).

Let the well be unperforated over a fractional length \( l_t \) from the top of the grid block and over a fractional length \( l_b \) from the bottom (where the fractional lengths are the actual lengths divided by the vertical thickness of the block). Thus the well is perforated over a fraction \( 1-l_t - l_b \) of the block thickness.

The scaled water saturation end points are derived as follows. SWMIN is the water saturation in the grid block when the water oil contact reaches the bottom of the perforations,

\[ SWMIN = l_b(1-S_{oover}) + (1-l_b)S_{wco} \]  

and SWMAX is the water saturation in the grid block when the contact reaches the top of the perforations,
Now consider the grid block when it has instead gas encroaching from the top. Above the gas oil contact the gas saturation is \(S_{ogc} - S_{wco}\). Below the contact it is reasonable to assume that solution gas has been released from the oil to form a critical gas saturation \(S_g = S_{gcr}\). The scaled gas saturation end points are thus derived as follows. \(SGMIN\) is the gas saturation in the grid block when the gas oil contact reaches the top of the perforations,

\[
SGMIN = l_t(1-S_{ogc} - S_{wco}) + (1-l_t)S_{gcr}
\]

Eq. 3.45

and \(SGMAX\) is the gas saturation in the grid block when the contact reaches the bottom of the perforations,

\[
SGMAX = (1-l_b)(1-S_{ogc} - S_{wco}) + l_bS_{gcr}
\]

Eq. 3.46

In gas-water problems where water encroaches from the bottom, the scaled water saturation end points are

\[
SWMIN = l_b(1-S_{gcr}) + (1-l_b)S_{wco}
\]

\[
SWMAX = (1-l_t)(1-S_{gcr}) + l_tS_{wco}
\]

Eq. 3.47
COMPRPL is used to rescale the fluid saturations at the well connections, to take account of the position of the perforations and the distribution of the phases within large grid blocks. The local grid well connections must already be defined with the keyword COMPDATL.

COMPRPL is followed by any number of records, described below, each record terminated by a slash (/). A record may be terminated early at any item; the remaining items will assume their default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

Note that, as with the COMPVEL keyword, the COMPRPL keyword cannot be used in non-VE runs with the Hysteresis option on if the imbibition and drainage table numbers are different in any connection.

The set of records must end with a blank record, containing only a slash (/).

Each record can contain some or all of the following items:

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Name of local grid containing the connections specified in this record
   If the local grid containing the well is not amalgamated (using keyword AMALGAM), this item may be defaulted or set to a blank string. The local grid name then defaults to the well’s local grid named in keyword WELSPECL.

3. I - location of connecting grid block(s)
   DEFAULT: 0 (allows any I-value)

4. J - location of connecting grid block(s)
   DEFAULT: 0 (allows any J-value)

5. K - location of upper connecting block in this set of data
   DEFAULT: 0 (corresponds to top connection of well)

6. K - location of lower connecting block in this set of data
   DEFAULT: 0 (corresponds to bottom connection of well)

   The following rescaling quantities are applied to all the connections in the well which match the location indices specified in items 2 - 5. But if a location index is defaulted to zero, it plays no part in selecting which connections use the rescaling data. Thus if the I and J location indices are defaulted to zero, the rescaling data is applied to all connections in the well that are located between layers K1 and K2 specified in items 4 and 5. If all four location indices are defaulted, the rescaling data is applied to all the connections in the well.

7. Saturation table number for connection relative permeabilities
   If the number is zero or positive, it will override the saturation table number specified in item 7 of keyword COMPDAT. If the number is zero, the relative permeabilities will be calculated using the same saturation table as the grid block containing the connection.
If the number is negative, the saturation table number will not be altered.

In runs using the hysteresis option (keyword **SATOPTS** in **RUNSPEC**), the saturation table number entered here will apply to both imbibition and drainage processes. Rescaling is not allowed if the imbibition and drainage table numbers for the connection are different.

**DEFAULT:** Negative

8. **SWMIN** Scaled value of the connate water saturation
   - The value is ignored in problems not containing water.
   - **DEFAULT:** $S_{wco}$

9. **SWMAX** Scaled value of the maximum water saturation
   - The value is ignored in problems not containing water.
   - **DEFAULT:** $1 - S_{owcr}$ in three-phase or oil-water problems, or $1 - S_{gcr}$ in gas-water problems.

10. **SGMIN** Scaled value of the critical gas saturation
    - The value is ignored in problems not containing both gas and oil.
    - **DEFAULT:** $S_{gcr}$

11. **SGMAX** Scaled value of the maximum gas saturation
    - The value is ignored in problems not containing both gas and oil.
    - **DEFAULT:** $1 - S_{ogcr} - S_{wco}$

**Example**

This example scales all connections in well PROD1 located in the local grid LGR1, and scales layer-2 connections in the local grid LGR2 for all wells with names beginning with P.

```
COMPRPL
  PROD1 LGR1 4* 3 .39 .63 .15 .29 /
  'P*' LGR2 2* 2 2 1* .39 .63 .15 .29 /
/
```

See the **COMPRP** keyword for more details on the saturation scaling calculation.
COMPS

Requests compositional mode

This keyword activates the compositional mode, and is an alternative to BLACKOIL. If neither COMPS nor BLACKOIL are specified, black oil mode will be assumed.

The COMPS data item specifies the number of components to be used in the simulation. In the documentation, the item in the COMPS keyword is frequently referred to as $N_c$.

Example

```
COMPS 12 /
```
**COMPSEGL**  Defines location of multisegment well completions in a local grid

This keyword defines the location of the completions in a multisegment well (see "Multisegment wells" in the ECLIPSE Technical Description), when the well is situated in a local grid, and instructs ECLIPSE to allocate each completion to a well segment. The well’s segment structure must previously have been defined with the WELSEGS keyword. Note that there is no equivalent WELSEGL keyword, as the segment structure is defined independently of the grid.

The location of the well connection to each completed grid block is defined within the well by the branch number and the length down the tubing to the start and end of the connection. This information must be supplied for all the well’s connections that have been specified with the COMPDATL keyword. ECLIPSE allocates each grid block connection to the segment whose nodal point lies nearest to the center of the connection (unless explicitly overridden by a user-specified segment). A segment (or to be more precise, its node) can accept flow from any number of grid-block connections. However, it is not possible to allocate a single grid-block connection to more than one segment.

The keyword must be used instead of COMPSEGS if the well is completed in one or more local grids that have been amalgamated. Either keyword may be used for wells completed in a single unamalgamated local grid.

The keyword can only be used on one well at a time, thus it must be specified more than once if there is more than one multisegment well.

The keyword is followed by a number of data records to define the location of all the grid block connections in the well. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

**Record 1**

The data for record 1 is identical to that of the COMPSEGS keyword.

**Each subsequent record**

The data for the subsequent records is the same as COMPSEGS except that there is an additional item at the start of each record:

1. Name of local grid containing the connections or range specified in this record

   This item may be defaulted or set to a blank string (provided that the local grid containing the well is not amalgamated in ECLIPSE 100). The local grid name then defaults to the well’s local grid named in keyword WELSECL.

   The connection locations in items 2, 3, 4 and 9 refer to their coordinates within this local grid.

The remaining items of these records (2 to 12) are the same as items 1 to 11 in COMPSEGS.

**Example**

A multilateral well with connections in two horizontal lateral branches in individual amalgamated local grids.

```
<table>
<thead>
<tr>
<th>Name</th>
<th>Local</th>
<th>I</th>
<th>J</th>
<th>K</th>
<th>Brn</th>
<th>Start</th>
<th>End</th>
<th>Dirn</th>
<th>End</th>
<th>Connection</th>
</tr>
</thead>
<tbody>
<tr>
<td>PROD</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
```

**Keywords**
<table>
<thead>
<tr>
<th>Grid</th>
<th>No</th>
<th>Length</th>
<th>Length</th>
<th>Penet</th>
<th>Range</th>
<th>Depth</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top Branch</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>UPPER</td>
<td>2</td>
<td>3</td>
<td>3</td>
<td>2</td>
<td>30</td>
<td>1*</td>
</tr>
<tr>
<td>Bottom Branch</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>LOWER</td>
<td>2</td>
<td>3</td>
<td>3</td>
<td>3</td>
<td>230</td>
<td>1*</td>
</tr>
</tbody>
</table>
COMPSEGS  Defines location of completions in a multisegment well

This keyword defines the location of the completions in a multisegment well (see "Multisegment wells" in the ECLIPSE Technical Description), and instructs ECLIPSE to allocate each completion to a well segment. The well’s segment structure must previously have been defined with the WELSEGS keyword. The location of the well connection to each completed grid block is defined within the well by the branch number and the length down the tubing to the start and end of the connection. This information must be supplied for all the well’s connections that have been specified with the COMPDAT (or COMPDATL) keyword. ECLIPSE allocates each grid block connection to the segment whose nodal point lies nearest to the center of the connection (unless explicitly overridden by a user-specified segment). A segment (or to be more precise, its node) can accept flow from any number of grid-block connections. However, it is not possible to allocate a single grid-block connection to more than one segment.

The keyword can only be used on one well at a time, thus it must be specified more than once if there is more than one multisegment well.

If the well is completed in one or more local grids that have been amalgamated, the keyword COMPSEGL must be used instead of COMPSEGS. Either keyword may be used for wells completed in single unamalgamated local grids.

If the well is completed in a local grid, the keyword COMPSEGL must be used instead of COMPSEGS.

The keyword is followed by a number of data records to define the location of all the grid block connections in the well. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

The data items in each record are:

**Record 1**

The first record consists of a single item of data that identifies the well.

1. Name of the well

End the data record with a slash (/).

**Each subsequent record**

The subsequent records set the start and end of each connection’s perforations in terms of distance along the tubing from the well’s zero tubing length reference point, and the branch number. The zero tubing length reference point is defined relative to the nodal point of the top segment by item 3 of the first record of keyword WELSEGS. The connections must previously have been defined with keyword COMPDAT (or COMPDATL as required).

The connections may either be specified individually, or in one or more “ranges”. A “range” is a contiguous row or column of grid blocks that are fully penetrated by the same branch of the well. The well is assumed to penetrate a “range” perpendicularly through the center of each grid block in the specified direction. The distance along the tubing to the start and end of each connection in the range is determined by summing the DX, DY or DZ thicknesses of the grid blocks in the range. All connections in the well must have their tubing distances set, either individually or within a range, working outwards along the main stem and each branch towards its “toe”.

1. I - location of the individual connection, or the connection at the start of the range (nearest the tubing head)
2. J - location of the individual connection, or the connection at the **start** of the range (nearest the tubing head).

3. K - location of the individual connection, or the connection at the **start** of the range (nearest the tubing head).

4. Branch number on which this connection (or range of connections) is located.

5. Distance down the tubing from the well’s zero tubing length reference point to the **start** of the connection in **this grid block**.
   
   **UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT - M)
   
   **DEFAULT:** Zero, for the first of these records, or the distance to the end of the previous connection or range, for subsequent records.

6. Distance down the tubing from the well’s zero tubing length reference point to the **end** of the connection in **this grid block**.
   
   This item, if set, must be greater than item 5.
   
   **UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT - M)
   
   **DEFAULT:** Value of item 5 + thickness of the grid block in the direction of penetration.

7. Direction of penetration through the grid block or the range.
   
   X or I  for horizontal penetration in the x-direction
   
   Y or J  for horizontal penetration in the y-direction
   
   Z or K  for vertical penetration
   
   This item is used for calculating the distance to the end of the perforations in the connection (or the connections in the range), where it defines which component of the grid block size to add (DX, DY or DZ). The item must be set if item 6 is defaulted, or if a range is specified in item 8. Otherwise it may be defaulted as it is not required.

8. Grid block I, J, or K coordinate at the **end** of the range, depending on the direction of penetration set in item 7. (For example the I-coordinate of the grid block at the end of the range, if the branch penetrates in the I-direction.)
   
   This may be greater than, or less than, the corresponding grid block coordinate in item 1, 2 or 3, depending on whether the branch penetrates out towards the positive or negative I, J or K direction.
   
   If this item is set, the tubing lengths are calculated for all the well’s connections that lie within the range. Should the range pass through an inactive grid block ECLIPSE prints a warning message and assume a zero thickness for that cell. If this is not appropriate, the range should be split and the thickness of the inactive grid block accounted for by explicitly defining the tubing lengths in items 5 and 6.
   
   If this item is defaulted, this record sets the tubing lengths for the single connection identified by items 1, 2 and 3.

9. Depth of the well connections within the range, that is the depth of the center of the perforations within each grid block in the range.
   
   If a default (or zero) value is entered, connection depths are calculated by linear interpolation from the segment data, according to the depths of the beginning and end of the segment and the position of the perforations within it.
If a negative value is entered, the connection depths remain at their original values, which are set equal to the center depths of the corresponding grid blocks when the connections are first defined with keyword COMPDAT.

ECLIPSE calculates the pressure difference between the connection and its corresponding segment from the fluid density within the segment, and adjusts the grid block pressure to the depth of the connection by adding a hydrostatic correction based on an average of the mobile fluid densities within the block (see “Inflow performance” in the ECLIPSE Technical Description).

See also item 63 of the OPTIONS keyword.

ECLIPSE 100

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: Zero, so that the depths are calculated from the segment data.

ECLIPSE 300 Thermal

10. Thermal contact length, that is, the length of the well in the completion cell.

This length is used with the specific thermal resistance, defined in WSEGHEAT, to calculate the conductive heat transfer between the well and the reservoir.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: The thickness of the grid block in the direction of penetration.

11. Segment number to allocate to all connections within the range.

If this record is negative or defaulted, then ECLIPSE allocates each grid block connection to the segment whose nodal point lies nearest to the center of the connection. If a segment number is provided then it is used instead - regardless of the physical tubing length positions of the segment and the connection.

DEFAULT: 0 (that is let ECLIPSE allocate the segment)

Note: If the segment number is specified using this item then the use of defaulting for the depth of the well connections (item 9 of this keyword) should be avoided. This is particularly true if the distance down the tubing to the middle of the connection does not lie within the tubing range of the specified segment (as the ECLIPSE calculation may then result in connections depths outside the reservoir).

End the data record with a slash (/).

End the set of data records with a slash.

Notes

Multilateral well data can be complicated to enter. It is advisable to check your data by examining the output generated in the Print file.

ECLIPSE 100

The output is generated by setting RPTSCHED keyword switch 14 (mnemonic WELSPECS).

ECLIPSE 300

The output is generated by the RPTPRINT keyword switches 6 and 7. These activate the WELL and COMPLETION summaries. A new segment summary is given under the WELL report in addition to a multisegment well report, and a new completion summary is given when a COMPLETION report is specified.

Example

A multilateral well with connections in three horizontal lateral branches.
## COMPSEGS

<table>
<thead>
<tr>
<th>Name</th>
<th>PROD</th>
<th>I</th>
<th>J</th>
<th>K</th>
<th>Brn</th>
<th>Start</th>
<th>End</th>
<th>Dirn</th>
<th>Penet</th>
<th>Range</th>
<th>Depth</th>
<th>Connection</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Top Branch</td>
<td>2</td>
<td>5</td>
<td>2</td>
<td>2</td>
<td></td>
<td>30</td>
<td>1*</td>
<td>X</td>
<td>6</td>
<td>6</td>
<td></td>
<td>/</td>
</tr>
<tr>
<td>Middle Branch</td>
<td>2</td>
<td>5</td>
<td>5</td>
<td>3</td>
<td></td>
<td>170</td>
<td>1*</td>
<td>X</td>
<td>6</td>
<td>6</td>
<td></td>
<td>/</td>
</tr>
<tr>
<td>Bottom Branch</td>
<td>2</td>
<td>5</td>
<td>8</td>
<td>4</td>
<td></td>
<td>230</td>
<td>1*</td>
<td>X</td>
<td>6</td>
<td>6</td>
<td></td>
<td>/</td>
</tr>
</tbody>
</table>
COMPSOL  Sets number of components in a SOLID, BLACKOIL run

This keyword is only necessary in a SOLID, BLACKOIL run. It is used to set the number of components.

The COMPSOL data item specifies the number of components to be used in the simulation. In the documentation, the item in the COMPSOL keyword is frequently referred to as $N_c$.

**Example**

```
COMPSOL
12 /
```

---

**Keywords**

519
**Total composition with respect to depth**

The data comprises a table composition with respect to depth data for each equilibration region. Each table consists of \( N_c + 3 \) columns of data, and is terminated by a slash (/).

| Column 1 | Depth values.  
|----------|-----------------  
|          | The values should increase monotonically down the column.  
|          | UNITS: \( m \) (METRIC), \( ft \) (FIELD), \( cm \) (LAB), \( m \) (PVT-M)  
| Columns 2 to \( N_c + 1 \) | The corresponding values of \( z_i \), the total phase mole fractions. The \( N_c \) mole fractions specified must add up to unity, and the program will check that this is the case.  
| Column \( N_c + 2 \) | Either 0 (vapor) or 1 (liquid) indicating that the fluid at this depth is above or below the gas-oil contact depth, respectively.  
| Column \( N_c + 3 \) | Observed saturation pressure, \( P_{\text{sat}} \), at this depth.  
|          | UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)  

The \( P_{\text{sat}} \) values are reported by the program in the solution versus depth output, for comparison with calculated values. They may be set to 0 if not required.

For depth values outside the interval specified, constant value extrapolation is performed.

The default maximum number of lines of COMPVD data is 50; this may be reset using the third item of keyword EQLDIMS.

For cases in which there is an initial gas-oil contact, compositions above the contact are taken as vapor compositions, and those below as liquid compositions. This is checked against the item value in column \( N_c + 2 \) to confirm that gas compositions have been entered above the gas-oil contact, and oil compositions below.

See also "Initial composition with respect to depth" in the ECLIPSE Technical Description.

**Example**

In this case a gas composition is specified at depths 4200, 4260 and 4350 feet, and a liquid composition at 4410 feet. This specifies the gas composition at the GOC of 4348 feet. Using compositional equilibration option 2 (EQUIL item 10), the program obtains the oil composition at the GOC from the gas dew point. Oil compositions below the GOC are interpolated from the table.

```
COMPVD
4200 .0121 .0194 .6399 .0869 .0591 .0967 .0474 .0331 .0053 0 3525
4260 .0121 .0194 .6499 .0869 .0591 .0967 .0474 .0241 .0043 0 3534
4350 .0121 .0194 .6599 .0869 .0591 .0967 .0474 .0151 .0033 0 3545
4410 .0114 .0148 .5517 .0873 .0653 .1299 .0800 .0401 .0191 1 3523
/  
```
COMPVE is used to reset the top and bottom depths of well connections, to allow for partial penetration within a grid block when calculating the phase relative permeabilities at the connection. The skin factor can also be adjusted to allow for partial penetration.

In runs using the vertical equilibrium option (keyword `VE` in the `RUNSPEC` section), the relative permeabilities at a well connection depend upon the connection's top and bottom depths and the fluid contact depths. Unless modified with the `COMPVE` keyword, the top and bottom depths of each connection are set to the depths of the center of the top and bottom faces of the connecting grid block, giving full penetration of the grid block.

If the `COMPVE` keyword is entered when the vertical equilibrium option is not in use, the top and bottom depths are automatically converted into scaled saturation table endpoints, which are then treated as if they had been entered with the `COMPRP` keyword. The conversion is performed by assuming a VE-type segregation of phases within the grid block (see “Representing the effects of partial penetration” in the `COMPRP` keyword). Note that, as with the `COMPRP` keyword, the `COMPVE` keyword cannot be used in non-VE runs with the Hysteresis option on if the imbibition and drainage table numbers are different in any connection. There is also a restriction on using `COMPVE` in non-VE runs that are ‘fast’ restarts (see keyword `RESTART`). The required grid block geometry data are only stored on the `SAVE` file if the base run contained keyword `COMPVE`. So if you wish to use `COMPVE` during a subsequent ‘fast’ restart, you must include this keyword in your base run, followed by just a slash if you don’t need to set any partial penetration data at that time. The restriction does not apply to flexible restarts, as the required data is generated from the `GRID` section.

When using the compressed vertical equilibrium option (item `COMPRESS` in `RUNSPEC` section keyword `VE`), each lumped column of cells may have several connections. If `COMPVE` is used for a vertical well in each of these connections, then the well have several distinct perforations in the lumped column of cells.

`COMPVE` is followed by any number of records, described below, each record terminated by a slash `/`. A record may be terminated early at any item; the remaining items will assume their default values. The set of records must end with a blank record, containing only a slash `/`.

Each record can contain some or all of the following items:

1. **Well name, well name template, well list or well list template**
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See ”Well name and well list template matching” in the `ECLIPSE Technical Description` for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword `WLIST`.

2. **I - location of connecting grid block(s)**
   DEFAULT: 0 (Allows any I-value)

3. **J - location of connecting grid block(s)**
   DEFAULT: 0 (Allows any J-value)

4. **K - location of upper connecting block in this set of data**
   DEFAULT: 0 (Corresponds to top connection of well)

5. **K - location of lower connecting block in this set of data**

6. **Saturation table number for connection relative permeabilities**
The end-points of the specified saturation table is used to set the relative permeabilities in each saturation zone, to obtain the fluid mobilities at the well connection. If the table number is zero or positive, it overrides the saturation table number specified in item 7 of keyword COMPDAT. If the number is zero, the fluid mobilities at the connection are calculated using the same saturation table as the grid block containing the connection.

DEFAULT: Negative.

7. **CVEFRAC** Fraction of vertical equilibrium curves to be used in calculating relative permeabilities for the connection.

   If **CVEFRAC** is 1.0, relative permeabilities are derived using a vertical equilibrium model, replacing the rock curve that applies to the connection. This assumes fully segregated flow into the connection. The end point values used in this Vertical Equilibrium model is taken from the table used for this connection, which may have been specified directly in item 6 above.

   If **CVEFRAC** is 0.0, the rock curve alone is taken: this should be used if a well pseudo has been generated, or if fully dispersed flow into the well is to be assumed. Again, the table used may have been specified directly in item 6.

   If **CVEFRAC** takes a value between 0.0 and 1.0, a combination of the two above extreme cases results: a fraction (1.0 - **CVEFRAC**) of the rock curve and **CVEFRAC** of the vertical equilibrium curve will be taken.

   If **CVEFRAC** < 0.0, the quantity is defaulted to the Vertical Equilibrium fraction for the grid block, specified in keyword VEFRAC. If the VEFRAC keyword was not specified, the default VEFRAC value of 1.0 is used.

   This item is ignored in non-Vertical Equilibrium runs.

   DEFAULT: Negative (The value is taken from VEFRAC)

8. **DTOP** The top depth of the top connection in the set matching the location indices in items 2 - 5.

   The value should not be less than the depth of the top-most corner of the connecting grid block, or the depth of the top of the grid block at the well’s position (set in item 12).

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

   DEFAULT: Depth of the center of the top face of the connecting grid block.

9. **DBOT** The bottom depth of the bottom connection in the set matching the location indices in items 2 - 5.

   The value should not exceed the depth of the bottom-most corner of the connecting grid block, or the depth of the bottom of the grid block at the well’s position (set in item 13).

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

   DEFAULT: Depth of the center of the bottom face of the connecting grid block.

10. Flag to recalculate the skin factor taking account of partial penetration

    **YES** The partial penetration skin factor will be calculated from Odeh’s formula (see Note below), overriding any value entered previously in item 11 of keyword COMPDAT. The connection transmissibility factor is then recalculated. The well bore diameter must be set in item 9 of keyword COMPDAT. The skin factor and transmissibility factor for the connection will also be recalculated each time there is a plug-back (see keyword WPLUG). Any skin due to other causes (for example damage or stimulation) should be entered in item 11 of this keyword.
Note that subsequent use of COMPDAT overrides the calculated skin with the value entered in COMPDAT item 11.

NO The skin factor and connection transmissibility factor will not be changed from their values set in COMPDAT.

DEFAULT: NO

11. \( S_d \) Skin factor due to damage (+ve) or stimulation (-ve).

This is used when recalculating the skin factor to take account of partial penetration, if item 10 above is set to YES. It is added to the partial penetration skin factor (see the note on calculation of the skin factor, below) to give the total mechanical skin. If the perforations penetrate the full thickness of the grid block, the partial penetration skin factor is zero and the total mechanical skin has the value entered here.

DEFAULT: 0.0

12. GTOP Depth of the top of the grid block at the position of the well connection. This is only relevant if the skin factor is to be recalculated for partial penetration, and the well is located in an off-center position in a sloping grid block.

The value should not be less than the depth of the topmost corner of the connecting grid block.

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: Depth of the center of the top face of the connecting grid block.

13. GBOT Depth of the bottom of the grid block at the position of the well connection. This is only relevant if the skin factor is to be recalculated for partial penetration, and the well is located in an off-center position in a sloping grid block.

The value should not exceed the depth of the bottommost corner of the connecting grid block.

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: Depth of the center of the bottom face of the connecting grid block.

**CAUTION:** If DTOP and DBOT are very close together, the fluid mobilities at the well connection are very sensitive to the contact depths or grid block saturations, which may cause convergence problems. This should be particularly noted when applying the keyword to horizontal wells.

**Example**

The first line of data sets the saturation table number, vertical equilibrium fraction, top-of-perforation depth, and bottom-of-perforation depth, for the connection(s) belonging to the well PROD1. The skin factor and connection transmissibility factor is recalculated to account for partial penetration. There is no skin due to damage.

The second line of data sets the top-of-perforation depth of the layer-3 connection of well PROD2. The skin factor and connection transmissibility factor remain unchanged at their values specified by the data in COMPDAT.

```
COMPVE
PROD1 4* 3 1.0 2010 2050 YES /
PROD2 2* 3 3 2* 2000 /
```
Calculation of the skin factor

If the completed length of the well does not cover the whole thickness of the formation, the convergence of the flow lines towards the perforated interval results in an additional component of the skin factor. For a well subject to both damage and partial penetration, the total mechanical skin factor is

\[ S = \frac{h_t}{h_p} S_d + S_p \]

where

- \( h_t \) is the thickness of the formation
- \( h_p \) is the length of the perforated interval
- \( S_d \) is the skin factor due to damage
- \( S_p \) is the skin factor due to partial penetration.

\( S_p \) is calculated from an equation given by Odeh (J. Pet. Tech., June 1980, p964):

\[ S_p = 1.35 \left( \frac{h_t}{h_p} - 1 \right)^{0.825} \left[ 1 \ln (h_{tpr} + 7) - 1 - 0.49 \ln (h_{tpr}) \ln (r_{wc}) \right] \]

where

- \( h_{tpr} = h_t \) divided by the square root of the vertical to horizontal permeability ratio \( K_v/K_h \)
- \( r_{wc} = r_w e^{0.216(z_m/(h_t+2.753))} \)
- \( z_m \) is the distance between the middle of the perforated interval and the top or bottom of the formation, whichever is nearer
- \( r_w \) is the wellbore radius.

All lengths are in feet.

The connection factor is then derived, using the calculated value of \( S \), as described in "The connection transmissibility factor" in the ECLIPSE Technical Description. Since the effects of partial penetration are contained in \( S \), the effective \( K_h \) value should be based on \( h_t \), not \( h_p \).

If the formation is divided vertically into two or more layers of grid blocks, the skin factor is calculated separately for each completed layer.

Thus

\[ h_t = GBOT-GTOP \]
\[ h_p = DBOT-DTOP \]
\[ z_m = \text{MIN}\{0.5(DTOP + DBOT)-GTOP, GBOT-0.5(DTOP + DBOT)\} \]

Keywords
**COMPVEL**

**Well connection depth data in local grids**

COMPVEL is used to reset the top and bottom depths of local grid well connections, to allow for partial penetration within a grid block when calculating the phase relative permeabilities at the connection. The skin factor can also be adjusted to allow for partial penetration.

In runs using the vertical equilibrium option (keyword **VE** in the **RUNSPEC** section), the relative permeabilities at the well connection depend upon the connection’s top and bottom depths and the fluid contact depths. Unless modified with the **COMPVEL** keyword, the top and bottom depths of each connection are set to the depths of the center of the top and bottom faces of the local grid block containing the connection, giving full penetration of the grid block.

If the **COMPVEL** keyword is entered when the vertical equilibrium option is **not** in use, the top and bottom depths are automatically converted into scaled saturation table endpoints, which are then treated as if they had been entered with the **COMPRPL** keyword. The conversion is performed by assuming a VE-type segregation of phases within the grid block (see “Representing the effects of partial penetration” in the **COMPRPL** keyword). Note that, as with the **COMPRPL** keyword, the **COMPVEL** keyword cannot be used in non-VE runs with the Hysteresis option on if the imbibition and drainage table numbers are different in any connection. There is also a restriction on using **COMPVEL** in non-VE runs that are ‘fast’ restarts (see keyword **RESTART**). The required grid block geometry data is only stored on the **SAVE** file if the base run contained either keyword **COMPVE** or **COMPVEL**. So if you wish to use **COMPVEL** during a subsequent ‘fast’ restart, you must include either of these keywords in your base run, followed by just a slash if you don’t need to set any partial penetration data at that time. The restriction does not apply to ‘flexible’ restarts, as the required data is generated from the **GRID** section.

**COMPVEL** is followed by any number of records, described below, each record terminated by a slash (/). A record may be terminated early at any item; the remaining items will assume their default values. The set of records must end with a blank record, containing only a slash (/).

Each record can contain some or all of the following items:

1. **Well name, well name template, well list or well list template**
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the **ECLIPSE Technical Description** for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword **WLIST**.

2. **Name of local grid containing the connections specified in this record**
   
   If the local grid containing the well is not amalgamated (using keyword **AMALGAM**), this item may be defaulted or set to a blank string. The local grid name then defaults to the well’s local grid named in keyword **WELSPECL**.

   The connection locations in items 3 - 6 refer to their coordinates within this local grid.

3. **I - location of connecting grid block(s)**
   
   DEFAULT: 0 (Allows any I-value)

4. **J - location of connecting grid block(s)**
   
   DEFAULT: 0 (Allows any J-value)

5. **K - location of upper connecting block in this set of data**
   
   DEFAULT: 0 (Corresponds to top connection of well)
6. K - location of lower connecting block in this set of data.  
   DEFAULT: 0 (Corresponds to bottom connection of well)  
   The following data quantities are applied to all the connections in the well that match the location indices specified in items 3 - 6. But if a location index is defaulted to zero, it plays no part in selecting the connections to which the following data applies. Thus if the I and J location indices are defaulted to zero, the data is applied to all connections in the well that are located between layers K1 and K2 specified in items 5 and 6. If all four location indices are defaulted, the data is applied to all the connections in the well.

7. Saturation table number for connection relative permeabilities.  
   The end-points of the specified saturation table are used to set the relative permeabilities in each saturation zone, to obtain the fluid mobilities at the well connection. If the table number is zero or positive, it overrides the saturation table number specified in keyword COMPDATL. If the number is zero, the fluid mobilities at the connection are calculated using the same saturation table as the grid block containing the connection.  
   If the number is negative, the saturation table number specified in keyword COMPDATL is used.  
   DEFAULT: Negative

8. CVEFRAC Fraction of vertical equilibrium curves to be used in calculating relative permeabilities for the connection.  
   If CVEFRAC is 1.0, relative permeabilities are derived using a vertical equilibrium model, replacing the rock curve that applies to the connection. This assumes fully segregated flow into the connection. The end point values used in this Vertical Equilibrium model are taken from the table used for this connection, which may have been specified directly in item 7 above.  
   If CVEFRAC is 0.0, the rock curve alone will be taken: this should be used if a well pseudo has been generated, or if fully dispersed flow into the well is assumed. Again, the table used may have been specified directly in item 7.  
   If CVEFRAC takes a value between 0.0 and 1.0, a combination of the two above extreme cases results: a fraction \(1.0 - CVEFRAC\) of the rock curve and CVEFRAC of the vertical equilibrium curve is taken.  
   If CVEFRAC < 0.0, the quantity is defaulted to the vertical equilibrium fraction for the grid block, specified in keyword VEFRAC. If the VEFRAC keyword was not specified, the default VEFRAC value of 1.0 is used.  
   This item is ignored in non-VE runs.  
   DEFAULT: Negative  
   (The value is taken from VEFRAC.)

9. DTOP The top depth of the top connection in the set matching the location indices in items 3 - 6.  
   The value should not be less than the depth of the top-most corner of the connecting grid block, or the depth of the top of the grid block at the well’s position (set in item 13).  
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)  
   DEFAULT: Depth of the center of the top face of the connecting grid block.

10. DBOT The bottom depth of the bottom connection in the set matching the location indices in items 3 - 6.
The value should not exceed the depth of the bottom-most corner of the connecting grid block, or the
depth of the bottom of the grid block at the well’s position (set in item 14).

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAult: Depth of the center of the bottom face of the connecting grid block.

11. Flag to recalculate the skin factor taking account of partial penetration

YES The partial penetration skin factor will be calculated from Odeh’s formula (see keyword
COMPVE), overriding any value entered in keyword COMPDATL. The connection
transmissibility factor is then recalculated. The well bore diameter must be set in keyword
COMPDATL. The skin factor and transmissibility factor for the connection is also recalculated
each time there is a plug-back (see keyword WPLUG). Any skin due to other causes (for
example damage or stimulation) should be entered in item 12 of this keyword.

Note that subsequent use of COMPDATL overrides the calculated skin with the value entered in
that keyword.

NO The skin factor and connection transmissibility factor is not changed from their values set in
COMPDATL.

DEFAult: NO

12. $S_d$ Skin factor due to damage (+ve) or stimulation (-ve).

This is used when recalculating the skin factor to take account of partial penetration, if item 11 above
is set to YES. It is added to the partial penetration skin factor to give the total mechanical skin. If the
perforations penetrate the full thickness of the grid block, the partial penetration skin factor is zero and
the total mechanical skin has the value entered here.

DEFAult: 0.0

13. GTOP Depth of the top of the grid block at the position of the well connection. This is only relevant if
the skin factor is to be recalculated for partial penetration, and the well is located in an off-center
position in a sloping grid block.

The value should not be less than the depth of the top-most corner of the connecting grid block.

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAult: Depth of the center of the top face of the connecting grid block.

14. GBOT Depth of the bottom of the grid block at the position of the well connection. This is only
relevant if the skin factor is to be recalculated for partial penetration, and the well is located in an off-
center position in a sloping grid block.

The value should not exceed the depth of the bottom-most corner of the connecting grid block.

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAult: Depth of the center of the bottom face of the connecting grid block.

**CAUTION:** If DTOP and DBOT are very close together, the fluid mobilities at the well connection are very
sensitive to the contact depths or grid block saturations, which may cause convergence problems. This
should be particularly noted when applying the keyword to horizontal wells.
See also the keywords `COMPVE`, `COMPRP` and `COMPRPL`.

**Example**

The first line of data sets the saturation table number, vertical equilibrium fraction, top-of-perforation depth and bottom-of-perforation depth, for the connection(s) belonging to the well PROD1 in the local grid LGR1. The skin factor and connection transmissibility factor will be recalculated to account for partial penetration. There is no skin due to damage.

The second line of data sets the top-of-perforation depth of the layer-3 connection of well PROD2 in the local grid LGR2. The skin factor and connection transmissibility factor remain unchanged at their original values.

```
COMPVEL
 PROD1 LGR1 4* 3 1.0 2010 2050 'YES' /
 PROD2 LGR2 2* 3 3 2* 2000 /
/}
```
**Requests multiple water component model**

This keyword activates the multiple water component model. It cannot be used with the `THERMAL` option. The `COMPW` data item specifies the number of water components to be used in the simulation. See "Water components" in the *ECLIPSE Technical Description* for further details.

**Example**

```
COMPW 2 /
```
**CONDFLTS**

**Specifies conductive faults**

The **CONDFLTS** keyword defines the geometry of the conductive faults. This facility is designed to help with the modeling of parts of the reservoir which can be regarded as being in vertical equilibrium (VE); for example, regions where the throughput is of one or more orders of magnitude faster than in other parts of the reservoir.

Whilst the order in which the grid blocks are entered is arbitrary, the code does internally nominate the first one to be the representative cell of the fault. If a well connects to the fault then this representative cell must also be a connection of the well. (It need not be open, of course.)

One conductive fault is specified in each **CONDFLTS** keyword.

The following restrictions apply:

- Only ECLIPSE 300 black oil models
- No **AIM** runs
- No **GASWAT** option
- No parallel runs
- No thermal runs.

The **CONDFLTS** keyword contains the following data, terminated by a slash (/):

1. Name of the conductive fault (up to 8 characters)
2. Type of solution method to be employed within the conductive fault
   - **VE**: the VE approximation will be applied within the fault
   - **OFF**: calculates the geometry but performs no calculations
   - **COARSE**: assumes the solution is identical in all cells of the fault (VE is not applied)

   **Note**: The following solution method options are also available, but are only recommended for the use of developers.

   End the line by a slash (/)

The following lines define the grid cells in the conductive fault. There can be as many lines as needed to define the region.

1. **I1** Lower I-coordinate of the box in the parent grid.
2. **I2** Upper I-coordinate of the box in the parent grid.
3. **J1** Lower J-coordinate of the box in the parent grid.
4. **J2** Upper J-coordinate of the box in the parent grid.
5. **K1** Lower K-coordinate of the box in the parent grid.
6. **K2** Upper K-coordinate of the box in the parent grid.
7. **ZGRID** Name of the parent grid.

   DEFAULT parent grid name: **GLOBAL**
Each line must be terminated by a slash (/).

**Note:** When using this keyword the conservation equations for all the cells are replaced with a conservation equation for the conductive fault. This is then solved for one pressure, water saturation and gas saturation. If using the VE option then the fluids will be redistributed among all the cells (water in the bottommost and gas in the topmost). However, there remains just the single pressure which can be a poor approximation in certain situations.

**Example**

```
CONDFLTS
   CFAULT1  VE /
   1 5 3 3 1 1 /
   3 3 3 3 2 10 /
/
```
CONDFRAC  Specifies a conductive fracture

The CONDFRAC keyword defines the geometry and properties of a single conductive fracture, to be modeled using the single medium conductive fracture formulation (see "Dual porosity" in the ECLIPSE Technical Description for further information.) The SCFDIMS keyword must be present to activate this feature. Multiple fractures may be defined using multiple instances of the CONDFRAC keyword.

The keyword should be followed by an initial record defining the fracture properties, and then a series of records describing the planar segments of the conductive fracture being modeled. There is no limit to the number of segments modeled. Note, however, that the total number of segments per cell modeled must not exceed the number declared in the SCFDIMS keyword.

Record 1

The initial record should contain the following four entries, terminated by a slash (/):

1. Name of the conductive fracture (up to 8 characters).
2. The saturation table number to use for the fracture.
3. The fracture effective aperture.
   
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
4. The fracture permeability.
   
   UNITS: mD (METRIC), mD (FIELD), mD (LAB), mD (PVT-M)

There are no defaults for these entries.

Subsequent records

The planar segment definitions should contain the following entries (again terminated by a slash).

1. I1 Lower I-coordinate of the segment in the global grid
2. I2 Upper I-coordinate of the segment in the global grid
3. J1 Lower J-coordinate of the box in the global grid
4. J2 Upper J-coordinate of the box in the global grid
5. K1 Lower K-coordinate of the box in the global grid
6. K2 Upper K-coordinate of the box in the global grid
7. FACE Segment orientation: ‘X’/’Y’/’Z’, or ‘I’,’J’,’K’.

Again, all information must be supplied - there are no defaults.

The whole of the keyword data should be terminated with a final slash (/).

Example

```
CONDFRAC
SCF1  2  1  1000 / 
6 6 4 5 1 4 'X' / 
6 7 5 5 1 4 'Y' / 
7 7 5 6 1 4 'X' / 
7 8 6 6 1 4 'Y' / 
```
<table>
<thead>
<tr>
<th>8 8 6 8 1 4 'X' /</th>
</tr>
</thead>
<tbody>
<tr>
<td>8 9 8 8 1 4 'Y' /</td>
</tr>
<tr>
<td>9 9 8 9 1 4 'X' /</td>
</tr>
</tbody>
</table>
The presence of the COORD keyword implies corner point geometry and the automatic generation of fault transmissibilities.

A coordinate line defines the possible positions for grid block corner points, for each (i,j) cell and for each reservoir in the grid. Given the depth of a particular grid block corner, and the associated coordinate line, the X and Y coordinates of the corner point can be calculated.

A coordinate line is specified by two triplets of X, Y and Z coordinates, representing two distinct points on it. If the (X, Y) coordinates of the top and bottom points are identical, then the Z coordinates of the points are not used.

The amount of data for this keyword can be quite voluminous, so it is recommended that a pre-processor (for example PETREL) be used to construct it.

If multiple reservoirs are present, this keyword must be preceded with keyword RESVNUM specifying the reservoir to which COORD data applies.

This keyword may not be used with block-centered geometry keywords, namely DR, DRV, DTHETA, DTHETAV, DX, DXV, DY, DYV, DZ, INRAD, TOPS.

The keyword line is followed by (NDIVIX + 1)(NDIVIY + 1)NUMRES coordinate lines, each specified by two points, each consisting of 3 values, namely, the X, Y and Z coordinates. NUMRES represents the number of reservoirs (or sets of coordinate lines) in the grid, which can be set with keyword NUMRES in the RUNSPEC section. NDIVIX and NDIVIY are specified in RUNSPEC keyword DIMENS.

The last coordinate line is followed by a slash (/).

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

Note: Theta values are always in degrees for radial runs.

Example

Here NDIVIX=3, NDIVIY=2 and NDIVI2=10, as specified in RUNSPEC keyword DIMENS, and NUMRES =1:

```
COORD
0  0    1000  0    0    1000
1000  0    1000  1000  0    1000
2000  0    1000  2000  0    1000
3000  0    1000  3000  0    1000
0    2000  1000  0    2000  1000
1000  2000  1000  1000  2000  1000
3000  2000  1000  3000  2000  1000
0    4000  1000  0    4000  1000
1000  4000  1000  1000  4000  1000
2000  4000  1000  2000  4000  1000
3000  4000  1000  3000  4000  1000
/```

Combining multiple reservoirs

Normally the data for the whole grid is specified in one COORD keyword. However, COORD data for individual sections of the grid can be entered in conjunction with the BOX keyword and/or the RESVNUM keyword, which is useful if you are combining separate models into a larger grid.
If just the BOX keyword has been entered (without RESVNUM), the program expects the following COORD keyword to contain the coordinate lines for the X-Y region within the box, for all NUMRES reservoirs. \((\text{NDX} + 1) \times (\text{NDY} + 1) \times \text{NUMRES}\) coordinate lines are expected, where \(\text{NDX}, \text{NDY}\) are the dimensions of the current box.

An easier way of combining multiple reservoirs is with the keyword RESVNUM, having set NUMRES in the RUNSPEC section equal to the number of reservoirs. The COORD data can be entered separately for each reservoir at a time, with the reservoir in question identified by the preceding RESVNUM keyword. Regions within each reservoir can be singled out with the BOX keyword. For a box of dimensions \(\text{NDX}, \text{NDY}\), the number of coordinate lines expected is \((\text{NDX} + 1) \times (\text{NDY} + 1)\).

If the separate reservoirs do not communicate with each other (in other words, they are coupled only via well group controls), they may be solved more efficiently as separate units if they are identified with the ISOLNUM keyword. Refer to "Independent reservoir regions" in the ECLIPSE Technical Description.
**COORDSYS  Coordinate system information for each reservoir**

This keyword supplies information about the coordinate system for each reservoir in the grid. For instance, it can assign layers of cells to multiple reservoirs.

This keyword is required only if the grid contains more than one reservoir (see keyword NUMRES). In ECLIPSE 100 the keyword may also be used if the grid completes the circle.

The keyword line is followed by NUMRES data records, each ending with a slash (/). Each record contains the following data items:

1. Lower bound for block index in K-direction (K1)
   For grid blocks in this reservoir (integer)

2. Upper bound for block index in K-direction (K2)
   For grid blocks in this reservoir (integer)

3. Completion of circle, for grid blocks in this reservoir
   
   **COMP**  Circle is completed in the Theta-(or Y-) direction
   **INCOMP** Circle is not completed

   For the circle to be completed, the coordinate lines to be identified must be coincident. In the case of cylindrical coordinates, this means that the Theta coordinates for nodes 1 and (NDIVIY + 1) in the Theta-direction must differ by 360 degrees

   **DEFAULT:** INCOMP

   **Note:** For radial local grids, circle completion (COMP) is the default - see "Local grid refinement and coarsening" in the *ECLIPSE Technical Description*.

4. Connection to the reservoir below

   Should be one of:

   **JOIN**  Transmissibilities are calculated to the reservoir below.

   **SEPARATE**  The reservoir is isolated from the reservoir below.

   This option calculates a transmissibility between cells (i,j,k) and (i,j,k+1) when the layers are in different reservoirs. This allows the description of bending coordinate lines. Care needs to be taken that the reservoirs coincide in a sensible manner. Note that fault non-neighbor connections are not calculated between different reservoirs.

   **DEFAULT:** SEPARATE

5. Lower bound for reservoir number for lateral connection to cells in different reservoirs (integer) (R1)

   **DEFAULT:** Current reservoir number

6. Upper bound for reservoir number for lateral connection to cells in different reservoirs (integer) (R2)

   **DEFAULT:** Current reservoir number

---

**Keywords**

536
The first two items of data for this keyword must have values which are less than or equal to the number of
global grid cells in the Z-direction (specified using item 3 in the `DIMENS` keyword in the `RUNSPEC`
section).

Items 5 and 6 may be used to allow fault NNCs between cells in different reservoirs. R1 and R2 should lie
between 1 and `NUMRES`, with `R1 \leq R2`. This allows a fault NNC to be calculated between cells (i,j,kl) in
reservoir number l, and (i+1,j,km) in reservoir m, if the limits `R1` and `R2` specified for reservoir l satisfy `R1
\leq m \leq R2`. This may be useful in accounting for 'bending' coordinate lines near faults. If horizontal
pinchouts are enabled by using `PINCHXY` in the `GRID` section, then fault NNCs may also be created
between different reservoirs across inactive columns of cells which are pinched-out horizontally.

The values of R1 and R2 should be consistent between different reservoirs: if the data record for reservoir l
allows lateral connection to reservoir m, then the values of R1 and R2 specified for reservoir m should also
allow lateral connection to reservoir l. The program issues an error message if this is not so.

If `NUMRES` is 1, the bounds of the reservoir defaults to the whole grid.

**Example**

Here `NDIVIX=5`, `NDIVIY=3`, `NDIVIZ=4`, as specified in `RUNSPEC` keyword `DIMENS`, and `NUMRES` =2.

```
<table>
<thead>
<tr>
<th></th>
<th>K1</th>
<th>K2</th>
<th>completed</th>
<th>join below</th>
<th>lateral limits</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>COORDSYS</code></td>
<td>1</td>
<td>2</td>
<td>COMP</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>4</td>
<td>INCOMP</td>
<td>/</td>
<td></td>
</tr>
</tbody>
</table>
```

`ECLIPSE Reference Manual`

<table>
<thead>
<tr>
<th>Keywords</th>
</tr>
</thead>
<tbody>
<tr>
<td>537</td>
</tr>
</tbody>
</table>
COPY

Copies data from one array to another

This keyword is used to assign or replace the value of a property for a box of cells within the grid, using the value of another property.

The keyword may be followed by any number of records, each of which is terminated by a slash (/). The data is terminated by a null record (that is, a record with no data before the terminating slash (/)).

Each record consists of at least two items of data:

1. **From** array.
   The name of the array to be used from which data is to be copied

2. **To** array.
   The name of the array to be set. Only this array is altered.
   The following data items may be used to redefine the input box for this and subsequent operations within the current keyword.
   
   If these items are not defined (a slash is inserted after item 2), they default to the values which were used for the previous operation within the current keyword.

   For the first operation in the keyword, the box defaults to the values set by the most recent **BOX** or **ENDBOX** keyword. If there is no preceding **BOX** or **ENDBOX** in the current section, the box is taken to include the entire reservoir.

3. First block to be modified on the X axis (IX1)

4. Last block to be modified on the X axis (IX2)

5. First block to be modified on the Y axis (JY1)

6. Last block to be modified on the Y axis (JY2)

7. First block to be modified on the Z axis (KZ1)

8. Last block to be modified on the Z axis (KZ2)

The data must satisfy:

\[ 1 \leq IX1 \leq IX2 \leq NDX \]
\[ 1 \leq JY1 \leq JY2 \leq NDY \]
\[ 1 \leq KZ1 \leq KZ2 \leq NDZ \]

where NDX, NDY, NDZ are the limits in the current **BOX**.

In the **REGIONS** section, it is possible to copy from the flux region array defined using the **FLUXNUM** keyword in the **GRID** section. This enables, for example, the **FIPNUM** array to be copied from the **FLUXNUM** array. Note that if part of the field is defined as flux region zero and subsequently copied into an array in the **REGIONS** section, the destination array will be set to 1.

Almost any array defined across the grid may be used with COPY. Examples are shown below:

In ECLIPSE 100:

This keyword may not be used with the **PBUB** or **PDEW** arrays as either **From** or **To**.

**Keywords**
Legal arrays

GRID section arrays

DX (DR)     DY (DTHETA)     DZ
PERMX (PERMR)  PERMY (PERMTHT)  PERMZ
MULTX (MULTR)  MULTY (MULTTHT)  MULTZ
PORO         NTG         DZNET
TOPS
DIFFMX (DIFFMR) DIFFMY (DIFFMTHT) DIFFMZ

ECLIPSE 300 only

MIDS

EDIT section arrays

PORV       DEPTH
TRANX (TRANR)  TRANY (TRANTHT)  TRANZ
DIFFX (DIFFR)  DIFFY (DIFFTHT)  DIFFZ
HEATTX (HEATTR)  HEATTY (HEATTHT)  HEATTZ (HEATTZ)
ROCKV

PROPS section arrays

SWL     SWCR     SWU
SGL     SGRU     SGRU
KRW     KRG     KRG
PCG     PCW

ECLIPSE 300 only

PSORG  PSORW  PSGRW
PSGRO  PSGWRO  PSGRGB
HSORW  HSWRGA  HSRCW
HSWRO  HSWRO  HSWRG

ECLIPSE 300 only

REGIONS section arrays

SATNUM  PVTNUM  EQLNUM
IMBNUM  FIPNUM  ENDMNUM
ROCKNUM  MISCNUM  SURFNUM
WH2NUM  WH3NUM

ECLIPSE 300 only
EOSNUM

ECLIPSE 300 only
PSTNUM  ISTNUM  DSTNUM

ECLIPSE 300 only
SDROW  SDROG  SDRGW

ECLIPSE 300 only
SDRWO  SDRGO  SDRWG

SOLUTION section arrays
PRESSURE  SWAT  SGAS
RV  RS  TBLK

ECLIPSE 100 only
GI  OILAPI  SALT

ECLIPSE 100 only
GASCONC  SOLVCONC  SOLVFRAC

ECLIPSE 100 only
SSOL

ECLIPSE 300 only
SOIL  PBUB  PDEW

ECLIPSE 300 only
TEMP  TEMPI

Notes

- The above list is not exhaustive, but includes the most common examples.
- Keywords in brackets are for radial geometry runs
- In the REGIONS section only integer values should be used
- Extreme caution should be exercised in using COPY for TRANX, TRANY and similar keywords: X- and Y-transmissibilities reflect cell positions, and have zero values at the edges of the grid

See also keywords BOX, ENDBOX, EQUALS, ADD and MULTIPLY.

Examples

Example 1

Set DY to DX, PERMY to PERMX:

COPY
DX    DY    /
PERMX PERMY /
/

Example 2

In the GRID section:

-------- SOURCE DESTINATION ------ BOX ------
COPY
PERMX PERMY  1  11  19  1  4  /

Keywords
540
PERMX PERMZ / defaults to last specified box
/
-------- ARRAY FACTOR
MULTIPLY
   PERMZ 0.1 /
/

In the REGIONS section:

COPY
   SATNUM FIPNUM /
/

Keywords

541
COPYBOX  

**Copies an array of grid data from one box to another**

The keyword may be followed by any number of records, each of which is terminated by a slash (/). The data is terminated by a null record (that is, a record with no data before the terminating slash).

Each record consists of the following items of data.

1. The name of the source array from which data is to be copied.
   - Items 2-7: Source box.
2. IX1S First block to be copied from on the X axis.
3. IX2S Last block to be copied from on the X axis.
4. JY1S First block to be copied from on the Y axis.
5. JY2S Last block to be copied from on the Y axis.
6. KZ1S First block to be copied from on the Z axis.
7. KZ2S Last block to be copied from on the Z axis.
   - Items 8-13 Destination box
8. IX1D First block to be copied to on the X axis.
9. IX2D Last block to be copied to on the X axis.
10. JY1D First block to be copied to on the Y axis.
11. JY2D Last block to be copied to on the Y axis.
12. KZ1D First block to be copied to on the Z axis.
13. KZ2D Last block to be copied to on the Z axis.

The data must satisfy:

\[
1 \leq IX1S \leq IX2S \leq NDIVIX \\
1 \leq IX1D \leq IX2D \leq NDIVIX \\
1 \leq JY1S \leq JY2S \leq NDIVIY \\
1 \leq JY1D \leq JY2D \leq NDIVIY \\
1 \leq KZ1S \leq KZ2S \leq NDIVIZ \\
1 \leq KZ1D \leq KZ2D \leq NDIVIZ (\text{where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS})
\]

The two boxes must be the same size in all three dimensions.

\[
IX2S - IX1S = IX2D - IX1D \\
JY2S - JY1S = JY2D - JY1D \\
KZ2S - KZ1S = KZ2D - KZ1D
\]

There must be no overlap between the two boxes.
Note: This keyword cannot be used if the PARALLEL keyword is present.

Examples

Example 1

Example 2

Keywords
**COPYREG**

**Copy by FLUXNUM region, MULTNUM region or OPERNUM region**

The keyword is followed by any number of records, each containing the items of data described below, and each terminated by a slash (/).

The records can be terminated early using a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. **From** array
   - The name of the array to be used.
2. **To** array
   - The name of the array to be set.
3. The flux region number, MULTNUM region number, or OPERNUM region number (with choice of region determined using item 4)
   - The flux region number refers to the region defined using the FLUXNUM keyword in the GRID section. It is not necessary to use the Flux Boundary option.
   - The MULTNUM region number refers to the region defined using the MULTNUM keyword in the GRID section.
   - The OPERNUM region number refers to the region defined using the OPERNUM keyword in the GRID or REGIONS section.
4. **Choice of region**: MULTNUM (M), FLUXNUM (F) or OPERNUM (O).
   - **DEFAULT**: M

*Note*: The FLUXNUM keyword, MULTNUM keyword or OPERNUM keyword must be entered before the COPYREG keyword. If item 4 is defaulted, then the MULTNUM keyword is used.

---

**LEGAL ARRAYS**

**GRID section arrays**

- DX (DR)
- DY (DTHETA)
- DZ
- PERMX (PERMR)
- PERMY (PERMTHT)
- PERMZ
- MULTX (MULTR)
- Multz (MULTTHT)
- MULTZ
- PORO
- NTG
- DZNET
- TOPS

---

**Keywords**
DIFFMX (DIFFMR) DIFFMY (DIFFMTHT) DIFFMZ

ECLIPSE 300 only

EDIT section arrays

PORV DEPTH

TRANX (TRANR) TRANY (TRANTHT) TRANZ

DIFFX (DIFFR) DIFFY (DIFFHTHT) DIFFZ

HEATTX (HEATTR) HEATTY (HEATHTHT) HEATTZ (HEATHTZ)

ROCKV

ROCKV

ECLIPSE 300 only

PROPS section arrays

SWL SWCR SWU

SGL SGCR SGU

KRW KRO KRG

PCG PCW

ECLIPSE 300 only

REGIONS section arrays

SATNUM PVNUM EQLNUM

IMBNUM FIPNUM ENDNUM

ROCKNUM MISCNUM WH2NUM

WH3NUM

EOSNUM

ECLIPSE 300 only

SOLUTION section arrays

PRESSURE SWAT SGAS

RV RS TBLK

ECLIPSE 100 only

GI OILAPI SALT

ECLIPSE 100 only

GASCONC SOLVCONC SOLVFRAC

ECLIPSE 100 only

SSOL

ECLIPSE 300 only

SOIL PBUB PDEW

ECLIPSE 300 only

TEMP TEMPI

Keywords

545
Notes

- The above list is not exhaustive, but includes the most common examples.
- Keywords in brackets are for radial geometry runs.
- Each record must refer to the same region. It is not possible to choose a different region for each data record.
- The COPYREG keyword is not available for use with local grid systems.

See also keywords ADDREG, EQUALREG, FLUXNUM, MULTNUM, OPERNUM and MULTIREG.

Examples

Example 1

Set PERMY to PERMX in flux region 3:

```
COPYREG
  PERMX PERMY 3 /
/
```

Example 2

Set DY to DX in MULTNUM region 1:

```
COPYREG
  DX DY 1 /
/
```
**COVTYPE**

**Component oil viscosity type**

This keyword can be used with the **THERMAL** option to specify how the oil viscosity is computed, such as using the Lorentz-Bray-Clark viscosities. With the keyword present an equation of state (EoS) can be used.

The keyword should be followed by one value for each component. The value should be one of the following:

<table>
<thead>
<tr>
<th>Type</th>
<th>Component</th>
</tr>
</thead>
<tbody>
<tr>
<td>NO</td>
<td>Not used. The viscosity for this component is computed as if this keyword was not present.</td>
</tr>
<tr>
<td>LMIX</td>
<td>The LBC viscosity model is applied with these components’ oil phase molar density.</td>
</tr>
<tr>
<td>LEMIX</td>
<td>The LBC viscosity model is applied on the LEMIX components densities where an EoS is used to compute the phase molar density.</td>
</tr>
</tbody>
</table>

*Table 3.6: COVTYPE component values*

If the keyword is present the default is **LMIX**.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword **TABDIMS**. The use of multiple EoS regions is discussed in "Equations of state regions" in the *ECLIPSE Technical Description*.

**Note:** If only a subset of the mixture is modeled by an EoS and many of the light components are included, and the heavy components are not, the mixture could yield a molar volume similar to that of an GAS and consequently the viscosity will be similar to that of a gas.

**Note:** Special care should be taken when the LBC viscosity model is applied to heavy components or gas condensate components.

For more information see **Thermal option** in the *ECLIPSE Technical Description*.

**Example**

In a thermal run with 4 hydrocarbon components:

```plaintext
COVTYPE
  LMIX LMIX LMIX LMIX /
```

**Keywords**

547
CPR Linear solver

This keyword activates the CPR linear solver. This alternative linear solver can often reduce simulation times for difficult problems, and is most likely to improve performance when the linear equations frequently fail to converge with the default linear solver. This situation can be identified from repeated linear convergence warnings output from the simulation. In these cases the CPR linear solver should be considered.

The CPR linear solver employs the constrained pressure residual (CPR) preconditioner, which provides a more robust method for solving the linear equations. For further details please refer to "The Constrained Pressure Residual (CPR) Preconditioner" in the ECLIPSE Technical Description.

The maximum number of linear iterations allowed for the outer iterations can be controlled with the TUNING keyword. By default, the maximum number of linear iterations allowed, in a Newton iteration, is 25 for ECLIPSE 100 and 20 for ECLIPSE 300.

The keyword does not contain any data items, and must be terminated by a slash (/).

In ECLIPSE 100, there are some options that cannot be used with the CPR linear solver. These include IMPES, the multiple independent reservoirs option, the Russian Doll option, the friction well option, triple porosity, nested local grid refinements, single phase simulations, item 1 of keyword WSEGSOLV, BTOBALFA, BTOBALFV and the rivers option.

In ECLIPSE 300, it is currently not possible to use the CPR linear solver for the thermal option, the multiple (including triple) porosity option, in a dual porosity (or dual permeability) run containing local grid refinements and the IMPES option.

In ECLIPSE 300, the CPR linear solver generally performs best when the simulation is fully implicit. For simulations that use the adaptive implicit (AIM) scheme, the performance may deteriorate if the fraction of fully implicit cells is small.

Notes

- Each linear iteration of the CPR linear solver takes more time, when compared to the default linear solver. If the CPR preconditioner is unable to reduce the total number of linear iterations significantly, this may cause the overall simulation time to increase when the CPR linear solver is used. For this reason, the CPR linear solver is best suited to cases which experience linear convergence problems with the default linear solver.
- The CPR linear solver cannot be expected to improve performance if the simulation is experiencing convergence problems due to highly non-linear behavior, which requires short time steps to resolve. In these cases the input data and existing convergence criteria should be checked, to ensure that the behavior is intended.

Example

```
CPR/
```

Keywords

548
**Oil component compressibility**

This keyword can only be used with the THERMAL option.

CREF specifies the liquid compressibility for each of the \( N_c \) hydrocarbon components.

The expression used to calculate oil component liquid densities is the following:

\[
\rho_c = \frac{\rho_{\text{ref},c}}{\left[1 - c_p \left( \frac{P - P_{\text{ref},c}}{P_{\text{ref},c}} \right) \right] \left[1 + c_{T1} \left( \frac{T - T_{\text{ref},c}}{T_{\text{ref},c}} \right) \right]}
\]

Eq. 3.51

where

- \( \rho_{\text{ref},c} \) is the reference density, specified with keyword DREF
- \( c_p \) is the oil component liquid compressibility, specified with this keyword
- \( P_{\text{ref},c} \) is the reference pressure specified, with keyword PREF
- \( T_{\text{ref},c} \) is the reference temperature, specified with keyword TREF
- \( c_{T1} \) is the thermal expansion coefficient, specified with keyword THERMEX1.

There should be one value for each component:

**UNITS:** 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

**DEFAULT:** 0.0

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

**Example**

For 3 components

```
CREF
  --Compressibility
  5.0E-06  5.2E-6  5.4E-6  /
```
**Water component compressibility**

CREFW specifies the water compressibility for each of the N\(_w\) water components.

The water component densities, at pressure \(P\), are calculated from:

\[
\rho_w = \rho_{refw} \cdot \left(1 + X + \frac{X^2}{2}\right)
\]

\[
X = c_{Pw} \cdot \left(P - P_{refw}\right)
\]

where

- \(\rho_{refw}\) is the reference density, specified with keyword **DREFW**
- \(c_{Pw}\) is the water component compressibility, specified with this keyword
- \(P_{refw}\) is the reference pressure specified, with keyword **PREFW**

There should be one value for each component:

- **UNITS:** 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)
- **DEFAULT:** 4.0e-5 1/atm

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of **TABDIMS**. The use of multiple EoS regions is discussed in "Equations of state regions" in the *ECLIPSE Technical Description*.

See "Water components" in the *ECLIPSE Technical Description* for further details of the water component option.

**Example**

For 2 components

```
CREFW
  --Compressibility
  5.0E-06  5.2E-6
```
CREFWS  Reference compressibility for water components for surface EoS

This optional keyword supplies reference compressibilities for water components for use in the surface calculations. The default, if this is not supplied, is for these to follow the reservoir values supplied with CREFW.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation-of-state regions is entered as item 10 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.
Permeability criterion for Vertical Equilibrium cell compression

The CRITPERM keyword may optionally be used in compressed vertical equilibrium runs (keyword VE in the RUNSPEC section). It specifies a critical permeability, above which cells can be collapsed, or merged with other cells in the same column, to form the larger compressed vertical equilibrium cells.

The keyword should be followed by a single real number that denotes the critical permeability for cell compression.

UNITS: mD (METRIC), mD (FIELD), mD (LAB), mD (PVT-M)

A cell is only collapsed into a larger compressed vertical equilibrium cell if the Z-direction permeability (entered using keyword PERMZ) exceeds this value.

If this keyword is not present, the critical permeability for cell compression defaults to 0.0, that is cells with Z-permeability > 0.0 can be collapsed.

A cell is merged with the next active cell below, provided that none of the following situations occur:

- An integer of 0 has been entered for that cell in keyword COLLAPSE
- TRANZ = 0.0
- PERMZ ≤ critical value entered in CRITPERM
- There is an inactive cell with DZ > 0.0 between the two active cells
- Either cell has been refined using RADFIN or CARFIN (see "Local grid refinement and coarsening" in the ECLIPSE Technical Description).

See also keyword COLLAPSE and "Compressed vertical equilibrium option" in the ECLIPSE Technical Description.

Example

```
CRITPERM
  0.2 ;
```
CRNDENS  Rock density by coal regions

The keyword is used to specify a coal density used in connection with a coal region number.

For a coal bed methane run, the density of the coal can be set constant for each coal region. This is an alternative to the ROCKDEN keyword specifying the rock density on a cell by cell basis. If both ROCKDEN and CRNDENS are specified, CRNDENS will only apply to cells undefined by ROCKDEN.

Each record consists of the following items:

1. The coal region number. This cannot be defaulted.
2. The corresponding rock/coal density.

    UNITS: kg/m$^3$ (METRIC), lb/ft$^3$ (FIELD), gm/cc (LAB), kg/m$^3$ (PVT-M)

Each record must be terminated by a slash(/), and the keyword is terminated by a second slash (/).

**Example**

```
CRNDENS
  1 89.5 /
  2 99.0 /
```

Keywords

553
CSKIN

Update well connection skin factors

CSKIN is used to update the skin factors for well connections. This subsequently changes the well connection transmissibility factors. The well connections must already be defined before this keyword can be used.

CSKIN is followed by any number of records, described below, each record terminated by a slash (/). A record may be terminated early at any item; the remaining items assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

Each record can contain some or all of the following items:

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. I - location of connecting grid block(s)
   DEFAULT: 0 (allows any I-value)

3. J - location of connecting grid block(s)
   DEFAULT: 0 (allows any I-value)

4. K - location of upper connecting block in this set of data
   DEFAULT: (corresponds to top connection of well)

5. K - location of lower connecting block in this set of data
   DEFAULT: (corresponds to bottom connection of well)

6. Connection skin factor. User defined arguments can be set with keywords UDQ and UDADIMS.

   Large negative skin factors can cause problems if they increase the effective wellbore radius to approach the pressure equivalent radius of the grid block. Very high values of the connection transmissibility factor may result, which could cause convergence problems. In these circumstances, it may be preferable to change the grid block data near the well to model the physical cause of the negative skin (for example enhance the permeability out to a certain distance from the well). If the effective wellbore radius is increased beyond the pressure equivalent radius, the connection factor becomes negative and an error message is issued.

   DEFAULT: 0.0

Note: Only one UDA may be defined for the skin factor per well at any one time. The skin factors may take different values for each connection but all must be calculated from the same UDQ formula. If two UDAs are defined for the skin factor for different connections on the same well then the skin factor for all UDA-controlled connections will be calculated using the most recently requested UDA.
Example

```
CSKIN
PROD1 4* CUSKN /
'p*' 2* 2 2 -1.5 /
/
```
Convergence criteria

This is an optional keyword which modifies the convergence criteria used to control the simulator in the SCHEDULE section. This keyword may be used more than once, for example to tighten up the criteria during a difficult part of the study.

The keyword may be followed by some or all of the following data items, terminated by a slash (/).

1. Maximum pressure change over an iteration.
   If the largest pressure change (for any cell) for a non-linear iteration is less than this value the solution is accepted. The solution obtained is usually at least as accurate as this value.
   This item is also used as the maximum pressure change over an iteration in the Multisegment Well Model. (ECLIPSE 300 only)
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0.1 atm

2. Maximum number of non-linear iterations.
   The timestep will be reduced if this number is exceeded before the non-linear equations are converged.
   DEFAULT: 20 (Fully implicit)
   15 (Fully implicit Thermal)
   6 (IMPES)
   12 (AIM or IMPSAT)

3. Required improvement in linear solver residual squared. This is the required reduction in RMS$^2$ and the RMS equations are given in ECLIPSE 300 in the “ECLIPSE Technical Description”.
   DEFAULT: 1.0E-10

4. Maximum number of linear iterations
   DEFAULT: 40, or NSTACK if NSTACK is greater than 40
   20, or NSTACK if NSTACK is greater than 20 if using the JALS solver or CPR solver

5. Maximum fugacity (phase equilibrium) error convergence criterion.
   (This option is not valid with the Thermal option.)
   DEFAULT: 0.001

6. Minimum number of linear iterations
   DEFAULT: 1 (Non-Thermal) 2 (Thermal)

7. Maximum component specific volume change over an iteration.
   If the largest change (for any cell and any component) for a non-linear iteration is less than this value the solution is accepted. This is effectively the contribution to the saturation change due to any component.
   DEFAULT: 0.01 (Non-Thermal) 0.2 (Thermal)

   Thermal only

8. Maximum temperature change over an iteration.
If the largest change for a non-linear iteration is less than this value the solution is accepted.

**UNITS:** K (METRIC), °R (FIELD), K (LAB), K (PVT-M)

**DEFAULT:** 10.0

9. Minimum number of non-linear iterations.

**DEFAULT:** 1

**Thermal**

**DEFAULT:** 2


**DEFAULT:** 1.0E+20

11. The tolerance for the maximum-norm of the non-linear residual, regardless of the solution change criteria. The solution is flagged as converged if the maximum-norm of the non-linear residual is less than this value, regardless of the solution change criteria. Please also refer to item 41.

**DEFAULT:** 1.0E-4

12. The tolerance for the sum of the non-linear residual, regardless of the solution change criteria. The solution is flagged as converged if the sum of the non-linear residual is below this value, regardless of the solution change criteria. Please also refer to item 42.

**DEFAULT:** 1.0E-20

13. Maximum allowed change in molar densities per non-linear iteration

**DEFAULT:** 2 x target value given by item 10 of TSCRIT

**Thermal option only**

14. Maximum allowed change in temperature per non-linear iteration

**UNITS:** K (METRIC), °R (FIELD), K (LAB), K (PVT-M)

**DEFAULT:** target value given by item 17 of TSCRIT

15. Maximum allowed change in pressure per non-linear iteration.

**UNITS:** bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

**DEFAULT:** Minimum of 100 atmospheres and 20% of the average pressure. Minimum of 100 atmospheres and the average pressure (Thermal)

16. Reserved for future use.

17. Reserved for future use.

18. Reserved for future use.

19. Reserved for future use.

20. Not used at present.

21. Not used at present.

22. Maximum $L_2^2$ solver residual norm.

**DEFAULT:** 1.0E-11

23. Maximum number of iterations for converging the Multisegment Well Model.

**DEFAULT:** 40

**Keywords**

557
   
   UNITS: kg-M/rm³ (METRIC), lb-M/rb (FIELD), gm-M/rcc (LAB), kg-M/rm³ (PVT-M)  
   DEFAULT: 0.015 kg-M/rm³

   
   UNITS: kg-M/day (METRIC), lb-M/day (FIELD), gm-M/hr (LAB), kg-M/day (PVT-M) 
   DEFAULT: 50 kg-M/day

   
   UNITS: kJ/rm³ (METRIC), Btu/rb (FIELD), J/rcc (LAB), kJ/rm³ (PVT-M) 
   DEFAULT: 50 kJ/rm³

   
   UNITS: m/s (METRIC), ft/s (FIELD), cm/s (LAB), m/s (PVT-M) 
   DEFAULT: 10 m/s

28. Convergence tolerance for rock displacement in the geomechanics calculation
   
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M) 
   DEFAULT: 0.01 m

29. Convergence tolerance for porosity in the geomechanics calculation
   
   UNITS: dimensionless 
   DEFAULT: 0.0001

30. Damping factor for partly coupled geomechanics
   
   DEFAULT: 1.0

31. Convergence tolerance for Multisegment Well Model maximum molar residual
   
   UNITS: kg-M/day (METRIC), lb-M/day (FIELD), gm-M/hr (LAB), kg-M/day (PVT-M) 
   DEFAULT: 1000 kg-M/day

32. Convergence tolerance for Multisegment Well Model maximum energy residual
   
   UNITS: kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB), kJ/day (PVT-M) 
   DEFAULT: 1E20 kJ/day

33. The SAMG item CHKTOL.
   
   Defines the amount of checking performed on the sparse matrix input to the SAMG solver. Negative values imply no checking, zero gives standard checking and positive values give enhanced checking. See also SAMG. 
   DEFAULT: - 1.0.

34. The SAMG item IFIRST.
   
   Defines the initial approximation used in the SAMG linear solver (1 for a constant zero vector, 2 for a constant vector of one, and 3 for a random vector). See also SAMG.
35. Convergence tolerance for Geomechanics finite element plasticity
   DEFAULT: 0.001 times the norm of the initial solution
36. Maximum number of iterations for Geomechanics finite element plasticity
   DEFAULT: 15
37. Maximum pressure change in Multisegment Well Model above which the solution vector will be dampened
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 250 atm
38. Maximum specific volume change in Multisegment Well Model above which the solution vector will be dampened.
   DEFAULT: 100
39. Maximum temperature change in Multisegment Well Model above which the solution vector will be dampened.
   UNITS: K (METRIC), °R (FIELD), K (LAB), K (PVT-M)
   DEFAULT: 200 K
40. Maximum fraction of Appleyard chop iterations per timestep.
   The Appleyard chop is used to stabilize the non-linear convergence. It modifies the Newton update in a cell, depending on phase appearance, phase disappearance and critical values of saturations. However, sometimes the Appleyard chop may actually prevent convergence. Therefore the Appleyard chop is switched off in a cell, during an iteration, if it has been performed more than $f \times MAXITS$ times during a timestep, where $MAXITS$ is defined by item 2 of this keyword and $f$ is defined by this item. If $f$ is set greater than 1, then the Appleyard chop will not be switched off, giving the pre-2011.1 behavior.
   DEFAULT: 0.5
41. Additional tolerance for the maximum-norm of the non-linear residual. The solution is only flagged as converged if it satisfies all other convergence criteria, including solution change, and the maximum-norm of the non-linear residual is less than this value.
   DEFAULT: 1.0E+20 (Not used)
42. Additional tolerance for the maximum sum of the non-linear residuals. The solution is only flagged as converged if it satisfies all other convergence criteria, including solution change, and the sum of the non-linear residual is less than this value.
   DEFAULT: 1.0E+20 (Not used)

**Example**

```
CVCRIT
  0.5 6 0.000001 20 /
```
Component volatility type

This keyword can be used with the **SOLID** option to identify solid phase components. It can also be used with the **THERMAL** option to specify whether a component exists only in the oil phase (a non-volatile component), or only in the gas phase (a non-condensable component).

The keyword should be followed by one value for each component. The value should be one of the following:

- **LIVE** for components that are both volatile and condensable.
- **SOLID** for components that are only in the solid phase.
- **DEAD** for components that are only in the oil phase.
- **GAS** for components that are only in the gas phase.

These values may be enclosed in quotes. Only the first character is significant. If this keyword is not used all components are treated as live components. The phases corresponding to each volatility type are summarized below:

<table>
<thead>
<tr>
<th>Type</th>
<th>Component</th>
<th>Option</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIVE</td>
<td>for components that are both</td>
<td></td>
</tr>
<tr>
<td></td>
<td>volatile and condensable.</td>
<td></td>
</tr>
<tr>
<td>SOLID</td>
<td>for components that are only in</td>
<td></td>
</tr>
<tr>
<td></td>
<td>the solid phase</td>
<td>Solid</td>
</tr>
<tr>
<td>DEAD</td>
<td>for components that are only in</td>
<td></td>
</tr>
<tr>
<td></td>
<td>the oil phase</td>
<td>Thermal</td>
</tr>
<tr>
<td>GAS</td>
<td>for components that are only in</td>
<td></td>
</tr>
<tr>
<td></td>
<td>the gas phase</td>
<td>Thermal</td>
</tr>
</tbody>
</table>

These values may be enclosed in quotes. Only the first character is significant. If this keyword is not used all components are treated as live components. The phases corresponding to each volatility type are summarized below:

<table>
<thead>
<tr>
<th>Type</th>
<th>Solid phase</th>
<th>Oil phase</th>
<th>Gas phase</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIVE</td>
<td></td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SOLID</td>
<td>x</td>
<td></td>
<td></td>
</tr>
<tr>
<td>DEAD</td>
<td></td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>GAS</td>
<td></td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

This keyword will override the critical temperature entered using the **TCRIT** keyword; for DEAD components the critical temperature is set to a large value ($10^4$); for GAS components the critical temperature is set to a small value ($10^{-6}$).

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword **TABDIMS**. The use of multiple EoS regions is discussed in Equations of state regions in the **ECLIPSE Technical Description**.

The solid option is described in Solid phase in the **ECLIPSE Technical Description**.

The thermal option is described in Thermal option in the **ECLIPSE Technical Description**.

Example

In a thermal run with 4 hydrocarbon components:

```plaintext
CVTYPE
GAS LIVE DEAD DEAD /
```
CVTYPES

Component volatility type for surface conditions

This keyword can be used with the SOLID option and the THERMAL option to specify the component volatility type at surface conditions. If this keyword is not used, the surface volatility type will be the same as the reservoir volatility type, set using the CVTYPE keyword.

The keyword is used in the same way as the CVTYPE keyword. It should be followed by one value for each component. The value should be one of the following:

<table>
<thead>
<tr>
<th>Type</th>
<th>Component</th>
<th>Option</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIVE</td>
<td>for components that are both volatile and condensable.</td>
<td></td>
</tr>
<tr>
<td>SOLID</td>
<td>for components that are only in the solid phase</td>
<td>Solid</td>
</tr>
<tr>
<td>DEAD</td>
<td>for components that are only in the oil phase</td>
<td>Thermal</td>
</tr>
<tr>
<td>GAS</td>
<td>for components that are only in the gas phase</td>
<td>Thermal</td>
</tr>
</tbody>
</table>

Table 3.9: CVTYPES component values

These values may be enclosed in quotes. Only the first character is significant. The phases corresponding to each volatility type are summarized below:

<table>
<thead>
<tr>
<th>Type</th>
<th>Solid phase</th>
<th>Oil phase</th>
<th>Gas phase</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIVE</td>
<td>x</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOLID</td>
<td>x</td>
<td></td>
<td></td>
</tr>
<tr>
<td>DEAD</td>
<td>x</td>
<td></td>
<td></td>
</tr>
<tr>
<td>GAS</td>
<td></td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

Table 3.10: Phases corresponding to each volatility type

Thermal only

This keyword will override the critical temperature entered using the TCRITS keyword; for DEAD components the critical temperature will be set to a large value (10^4); for GAS components the critical temperature is set to a small value (10^-6).

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in Equations of state regions in the ECLIPSE Technical Description.

The solid option is described in Solid phase in the ECLIPSE Technical Description.

The thermal option is described in Thermal option in the ECLIPSE Technical Description.

Example

In a thermal run with 4 hydrocarbon components:

```
CVTYPES
GAS LIVE DEAD DEAD /
```
Component water type

This keyword can be used with multi-component water (COMPW) to identify special water components. The keyword should be followed by one value for each component:

<table>
<thead>
<tr>
<th>Type</th>
<th>Component</th>
</tr>
</thead>
<tbody>
<tr>
<td>SURFF</td>
<td>Surfactant component for foam model (FOAMFSC)</td>
</tr>
<tr>
<td>SURFS</td>
<td>Surfactant component for surface tension effects (SURFST)</td>
</tr>
<tr>
<td>SURF</td>
<td>Surfactant component for both foam and surface tension effects</td>
</tr>
<tr>
<td>POLY</td>
<td>Polymer component for water phase viscosity effects</td>
</tr>
<tr>
<td>ALK</td>
<td>Alkaline component for water phase viscosity effects</td>
</tr>
</tbody>
</table>

*Table 3.11: CWTYPE component values*

All other water components should use the default type (for example 1*).

If multiple equations of state are being used, a set of data should be entered for each reservoir equation of state (EoS). The number of reservoir EoS regions is entered as the 9th parameter of keyword TABDIMS. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

Note that the multi-component water model is not currently compatible with THERMAL options, GASWAT option, GASSOL option, CO2SOL option, Satellite groups (see GSATPROD) and IMPSAT. See “Water components option limitations” in the ECLIPSE Technical Description.

**Examples**

**Example 1**

In a foam simulation with 2 water components:

```
CWTYPE
1* SURFF /
```

**Example 2**

In a foam and surfactant simulation with 3 water components:

```
CWTYPE
1* SURFF SURFS /
```

**Example 3**

In a foam and surfactant simulation with 2 water components:

```
CWTYPE
1* SURF /
```
This section contains an alphabetic listing of keywords starting with the letter D.
DATE

**Outputs the date to the summary file**

This keyword requests the date to be written to the Summary file. The date is stored as three summary mnemonics DAY, MONTH, YEAR.

The date is not usually used for plotting summary data (use TIME or YEARS); however, it is useful when the summary data is tabulated either in GRAF or by using the RUNSUM keyword.

The DATE keyword has no associated data.

*Note:* The DAY, MONTH and YEAR are stored as real numbers on the summary files but are output as integers in the tabulated output from GRAF or RUNSUM. In addition, if the DATE keyword is used in conjunction with RUNSUM, then the date instead of the time is printed in the left most column of the run summary.

**Examples**

**Example 1**

```plaintext
DATE
```

**Example 2**

Example RUNSUM output

<table>
<thead>
<tr>
<th>DATE</th>
<th>YEARS</th>
<th>DAY</th>
<th>MONTH</th>
<th>YEAR</th>
<th>FOPR</th>
<th>WGOR</th>
<th>WBHP</th>
</tr>
</thead>
<tbody>
<tr>
<td>19-OCT-92</td>
<td>0</td>
<td>19</td>
<td>10</td>
<td>1992</td>
<td>0</td>
<td>0</td>
<td>4800.000</td>
</tr>
<tr>
<td>20-OCT-92</td>
<td>0.002738</td>
<td>20</td>
<td>10</td>
<td>1992</td>
<td>20000.00</td>
<td>1.270000</td>
<td>2892.246</td>
</tr>
<tr>
<td>21-OCT-92</td>
<td>0.006462</td>
<td>21</td>
<td>10</td>
<td>1992</td>
<td>20000.00</td>
<td>1.270000</td>
<td>2743.021</td>
</tr>
<tr>
<td>24-OCT-92</td>
<td>0.014967</td>
<td>24</td>
<td>10</td>
<td>1992</td>
<td>20000.00</td>
<td>1.270000</td>
<td>2578.314</td>
</tr>
<tr>
<td>28-OCT-92</td>
<td>0.026090</td>
<td>28</td>
<td>10</td>
<td>1992</td>
<td>20000.00</td>
<td>1.270000</td>
<td>2459.743</td>
</tr>
<tr>
<td>3-NOV-92</td>
<td>0.043609</td>
<td>3</td>
<td>11</td>
<td>1992</td>
<td>20000.00</td>
<td>1.263817</td>
<td>2364.214</td>
</tr>
<tr>
<td>14-NOV-92</td>
<td>0.073839</td>
<td>14</td>
<td>11</td>
<td>1992</td>
<td>20000.00</td>
<td>1.241348</td>
<td>2287.771</td>
</tr>
</tbody>
</table>
DATES

Advances simulator to specified report date(s)

The keyword should be followed by a list of dates at which reports are required. Each date must be on a separate line terminated by a slash (/). The data is terminated by a null record (a slash on a line by itself).

A date consists of the following items of data:

1. The day of the month
   An integer between 1 and 31.

2. The name of the month abbreviated to three characters
   JAN, FEB, MAR, APR, MAY, JUN, JUL, AUG, SEP, OCT, NOV or DEC.
   JUL is an acceptable alternative to JLY.

3. The year
   A positive 4 digit integer.

4. The time
   A string of the format HH:MM:SS.SSSS.
   DEFAULT: 00:00:00

Note: The date at the start of the simulation (time 0) is entered using the keyword START in the RUNSPEC section.

Example

```
DATES
  1 FEB 1985 /
  1 MAR 1985 06:00:00 /
  1 MAY 1985 13:30:00 /
  1 JAN 1986 /
  1 JAN 1987 19:45:15.3333 /
/  
```
**DATUM**

**Datum depth for output of depth corrected pressures**

The keyword should be followed by a single positive real number, the datum depth for calculation of depth corrected pressures (or potentials), and a slash (/). DATUM is entirely optional. If it is not specified, and output of depth corrected pressures is requested, it defaults to the datum depth entered for equilibration region 1 using the **EQUIL** keyword, or to zero if there is no **EQUIL** keyword.

**UNITS:** m (**METRIC**), ft (**FIELD**), cm (**LAB**), m (**PVT-M**)

**Note:** For ECLIPSE 100 only, it is possible to specify a negative depth value.

**Example**

```
DATUM
4800 /
```
**DATUMR**

**Datum depth for each FIP region**

This keyword is optional and should be followed by the maximum number of fluid-in-place regions (NTFIP, see keyword REGDIMS or TABDIMS). Each number must be a positive real number, one for each FIP region in the model as defined by keyword FIPNUM, followed by a slash (/). The datum depths are used to calculate the depth corrected pressures (potentials).

If the DATUMR keyword is used then the datum specified in any DATUM keyword will be ignored.

If different datum depths are used in different regions the field average potentials (for example FPPO) have little physical meaning as cells are averaged based on different datum depths. Typically the field values should be ignored and the region average potentials utilized (for example RPPO).

**UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

---

**Note:** For ECLIPSE 100 only, it is possible to specify a negative depth value.

---

**Example**

```
DATUMR
4800 5000 5200 /
```
DATUMRX

Datum depth for each FIP region in each FIP set

This keyword is optional and similar to DATUM. However, it allows the provision of datum depths for all fluid-in-place sets defined by the FIP keyword, in addition to the standard set defined with keyword FIPNUM.

If the DATUMRX keyword is used then the datum specified in any DATUM keyword will be ignored.

Datum depths for all fluid-in-place sets must be specified unless the DATUMRX keyword has been processed previously, in which case the datum values of any unspecified fluid-in-place set will be based on the DATUMRX keyword.

**Note:** For ECLIPSE 100 only, an exception is the fluid-in-place set created by the keyword FIPWG whose datum depths will be identical to the depths of the default fluid-in-place unless explicitly provided.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. The fluid-in-place region family, to which the specified datum depths belong.
   - The family name should be up to 5 characters long.
   - A default (1*) or blank (' ') entry refers to the standard set of fluid-in-place regions defined with the keyword FIPNUM.
   - Additional families of fluid-in-place regions may optionally be defined, using the keyword FIP. The family name is a string of up to 5 characters that makes up the second segment of the keyword name (characters 4 - 8). Each family can divide up the field in different ways. Thus, for example, the standard FIPNUM family may divide the field into layers, and an additional family can be defined to divide the field into areal sectors.

2. The datum depths for each region of the fluid-in-place family.
   - A maximum number of NTFIP values is allowed as defined by keywords REGDIMS or TABDIMS. Each number must be a positive real number, one for each fluid-in-place region in the fluid-in-place.
   - UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   - If a datum value is not specified for all fluid-in-place regions, the datum values for the remaining fluid-in-place regions will be based on the last specified region.

The datum depths are used to calculate the depth corrected pressures (potentials). The datum depths provided for the fluid-in-place sets specified by the FIP keyword are used when calculating the SUMMARY keywords RPPO, RPPW, RPPG and RPPC (ECLIPSE 100 only).

**Note:** It is possible to specify a negative depth value.

**Example**

```
DATUMRX
'XXX' 4800 5000 5200 /
'1*' 4850 5100 /
/
```
DBGODD3P  Controls debug output for ODD3P option

This keyword can be used to write out debug information for the ODD3P relative permeability and capillary pressure hysteresis option. The data are output to the Debug file.

The keyword should be followed by a status string, controlling whether the debug output is selected or deselected, plus 6 integers, which define the box of cells for which output will be generated, terminated with a slash (/)

1. STATUS This should be one of two values:
   - ON  Switch debug output on
   - OFF Switch debug output off.

   **DEFAULT**: OFF

2. IX1 First cell on X axis to be included in output
3. IX2 Last cell on X axis to be included in output
4. JY1 First cell on Y axis to be included in output
5. JY2 Last cell on Y axis to be included in output
6. KZ1 First cell on Z axis to be included in output
7. KZ2 Last cell on Z axis to be included in output

For items 2 to 7, the values must satisfy:

\[1 \leq IX1 \leq IX2 \leq NDIVIX \]
\[1 \leq JY1 \leq JY2 \leq NDIVIY \]
\[1 \leq KZ1 \leq KZ2 \leq NDIVIZ \] (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS)

**Note:** It is recommended that this facility should be used with caution to avoid excessive output to the Debug file. The format and content of this debug output is subject to change.

---

**Example**

```
DBGODD3P
   -- STATUS IX1 IX2 IY1 IY2 KZ1 KZ2
   ON  4 4  5 5  1 3 /
```
Definition of the DCQ - gas rate or energy

This keyword may be used to specify whether the DCQ targets set in keyword GASYEAR, GASPERIO, GDCQ, GASFTARG or GASFDECR refer to energy or gas production rate. It is relevant only when running the gas field operations model in conjunction with the gas calorific value control option. (See "Gas field operations model" and "Gas calorific value control" in the ECLIPSE Technical Description for further details.)

The keyword should be followed by a single data item, terminated by a slash (/):

1. Quantity to which the DCQ refers.
   - GAS: The DCQ refers to the gas rate, which is standard for the Gas Field Operations Model.
   - ENERGY: The DCQ refers to the energy rate, and its target value should be entered in keyword GASYEAR, GASPERIO or GDCQ in the appropriate units: kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB), kJ/day (PVT-M). ECLIPSE give the contract groups a GCONENG target equal to the energy DCQ multiplied by the month’s swing or profile factor. Any economic DCQ limits set in keyword GDCQECON should also refer to energy rate.

   (The word may be abbreviated, as only the first character is significant.)

   DEFAULT: GAS.

End data record with a slash (/).

**CAUTION:** If the DCQDEFN keyword is used it must be entered before the keyword SWINGFAC or GSWINGF.

### Example

```
DCQDEFN ENERGY /
```
The dead oil option is to be used

This keyword can only be used with the THERMAL option (see "Thermal option" in the ECLIPSE Technical Description). It does not allow a hydrocarbon component to occur in the gas phase. Oil and water K-values do not need to be specified if this option is chosen. DEADOIL is an alternative to LIVEOIL and BLACKOIL.

This keyword has no associated data.

Note: This keyword cannot be used with more than one hydrocarbon component.
Controls on debug output

The `DEBUG` keyword is primarily intended for use as an aid to simulator development, and the function of its controls may change from version to version. Also there may be undocumented and limitations, without warnings, on its use. Debug output is not fully supported. The keyword is documented here for completeness.

Many of the controls will produce vast amounts of output and should be used with extreme caution.

The keyword should be followed by integers, each of which controls a particular form of debug output. A value less than or equal to zero switches the debug output off. To produce debug output from a particular option, the relevant debug switch should be set greater than zero. In some cases the volume of output depends on the value of the switch (in general larger values produce more output). The data field should be terminated by a slash (/). If fewer than the maximum number integers are read, the remainder are left unchanged. Once a particular debug item is activated, it remains activated in all subsequent uses of `DEBUG` keyword unless it is explicitly turned off.

Note: Repeat counts (for example 3*0) can be used if required (but no spaces before or after the asterisk).

<table>
<thead>
<tr>
<th>DEBUG item</th>
<th>Effect of DEBUG item</th>
</tr>
</thead>
<tbody>
<tr>
<td>Item 1</td>
<td>Terminal output copied to debug file. Timing reports if &gt; 1.</td>
</tr>
<tr>
<td>Item 2</td>
<td>Debug summary for all wells.</td>
</tr>
<tr>
<td>Item 3</td>
<td>Details of storage allocation.</td>
</tr>
<tr>
<td></td>
<td>If set &gt; 1 the local grid storage allocation is also output.</td>
</tr>
<tr>
<td>Item 4</td>
<td>Details of index arrays from RCINDX.</td>
</tr>
<tr>
<td>Item 5</td>
<td>Debug of environment variables.</td>
</tr>
<tr>
<td>Item 6</td>
<td>Debug from matrix set-up (see also switches 45-50).</td>
</tr>
<tr>
<td></td>
<td>If controls 45-50 are set to 0, then the matrix set-up debug is produced for the cells with the largest normalized residuals. The number of cells for which output is produced increases as the value of switch 6 increases.</td>
</tr>
<tr>
<td>Item 7</td>
<td>Debug from linear solver.</td>
</tr>
<tr>
<td>Item 8</td>
<td>Debug from equilibrator.</td>
</tr>
<tr>
<td>Item 9</td>
<td>Debug from non-linear solver (chops of (S_w, S_{ri}, S_{pp}, P_{o}, R_s, R_v, BHP)).</td>
</tr>
<tr>
<td></td>
<td>If &gt;1 debug of PVT extrapolation, and state transitions.</td>
</tr>
<tr>
<td>Item 10</td>
<td>Debug from NNC generation.</td>
</tr>
<tr>
<td>Item 11</td>
<td>Debug from network option balancing calculation.</td>
</tr>
<tr>
<td>Item 12</td>
<td>Debug from Fetkovich or Carter-Tracy aquifer calculations.</td>
</tr>
<tr>
<td>Item 13</td>
<td>Debug from vertical equilibrium option.</td>
</tr>
<tr>
<td>Item 14</td>
<td>Debug from transmissibility calculations.</td>
</tr>
<tr>
<td>Item 15</td>
<td>Debug from threshold pressure calculation.</td>
</tr>
<tr>
<td>DEBUG item</td>
<td>Effect of DEBUG item</td>
</tr>
<tr>
<td>------------</td>
<td>---------------------</td>
</tr>
<tr>
<td>Item 16</td>
<td>Debug from hysteresis option. If set &lt; 0, this removes the restriction on monotonically increasing relative permeability and rock compaction tables. If set &lt;-1. then, in addition, the restriction on monotonic capillary pressure functions is also removed. This is not recommended for general use because unphysical or non-monotone saturation functions will often cause severe convergence failures.</td>
</tr>
<tr>
<td>Item 17</td>
<td>Debug from API tracking option.</td>
</tr>
<tr>
<td>Item 18</td>
<td>Debug from the <strong>ACTION</strong> keyword.</td>
</tr>
<tr>
<td>Item 19</td>
<td>If &gt;0 grid displays of the non-linear residual function are produced in the debug file at each non-linear iteration.</td>
</tr>
<tr>
<td>Item 20</td>
<td>Debug from input parser and the I/O PUT and GET Routines.</td>
</tr>
<tr>
<td>Item 21</td>
<td>Debug from total compressibility check calculation.</td>
</tr>
<tr>
<td>Item 22</td>
<td>Debug from cascade solver.</td>
</tr>
<tr>
<td>Item 23</td>
<td>Debug from end-point scaling option.</td>
</tr>
<tr>
<td>Item 24</td>
<td>Debug from tracer tracking option.</td>
</tr>
<tr>
<td>Item 25</td>
<td>Debug from gravity imbibition and drainage calculation in dual porosity/dual permeability systems.</td>
</tr>
<tr>
<td>Item 26</td>
<td>This determines whether the checks specified by debug switches 6, 7, 27 and 38 apply to any local grids. When set to -1, all local grids are checked as well as the global grid; if set = 0, only the global grid is checked; if set = n&gt;0, then only the n&lt;sup&gt;th&lt;/sup&gt; local grid is checked.</td>
</tr>
<tr>
<td>Item 27</td>
<td>If &gt;0 Jacobian is checked by numerical differentiation of the residual, and any errors reported to the debug file. The cells checked are limited by switches 45-50.</td>
</tr>
<tr>
<td>Item 28</td>
<td>Debug from polymer flood model.</td>
</tr>
<tr>
<td>Item 29</td>
<td>Debug from dynamic memory allocation module.</td>
</tr>
<tr>
<td>Item 30</td>
<td>Debug from Gi-condensate/volatile oil model.</td>
</tr>
<tr>
<td>Item 31</td>
<td>Debug from gas field operations model.</td>
</tr>
<tr>
<td>Item 32</td>
<td>Debug from Solvent Model.</td>
</tr>
<tr>
<td>Item 33</td>
<td>Subroutine indicator to detect the position of a program hang.</td>
</tr>
<tr>
<td>Item 34</td>
<td>Debug of internal tables generated from alternative keywords.</td>
</tr>
<tr>
<td>Item 35</td>
<td>Debug from the surfactant model.</td>
</tr>
<tr>
<td>Item 36</td>
<td>Debug from nine-point scheme.</td>
</tr>
<tr>
<td>Item 37</td>
<td>Debug from gas lift optimization calculation.</td>
</tr>
<tr>
<td>Item 38</td>
<td>If &gt;0 a line search is performed along the newton direction. The following steps are used alpha = 10*0.1,-1,1. the search is only performed newton iteration &gt; debug(38).</td>
</tr>
<tr>
<td>DEBUG item</td>
<td>Effect of DEBUG item</td>
</tr>
<tr>
<td>------------</td>
<td>----------------------</td>
</tr>
<tr>
<td>Item 39</td>
<td>Debug for the parallel options, reservoir coupling and Open-ECLIPSE message passing routines. An output line for each message header is written to standard output for the master process. The equivalent slave data is written to a file called <code>kzpN.dbg</code>, where N is the process number. In addition consistency checking is performed between the header information and the subsequent message. Errors are written to the appropriate output streams if an inconsistency occurs.</td>
</tr>
<tr>
<td>Item 40</td>
<td>Debug from local grid refinement.</td>
</tr>
<tr>
<td>Item 41</td>
<td>Debug from parallel LGR option.</td>
</tr>
<tr>
<td>Item 42</td>
<td>Debug from reservoir coupling facility.</td>
</tr>
<tr>
<td>Item 43</td>
<td>Debug from the domain decomposition parallel option. When set = 2, if this is a distributed memory run the profile information for each slave task is output to the local debug file. The profile information is analogous to the standard profiling information output if the <code>CPU</code> mnemonic is used in the <code>RPTSCHED</code> keyword. The debug files have the form <code>root.n.dbg</code> where n is the slave number.</td>
</tr>
<tr>
<td>Item 44</td>
<td>Debug information about the phase flows.</td>
</tr>
<tr>
<td>Item 45</td>
<td>Lower bound on X axis of matrix set-up debug output (switch 6).</td>
</tr>
<tr>
<td>Item 46</td>
<td>Upper bound on X axis of matrix set-up debug output (switch 6).</td>
</tr>
<tr>
<td>Item 47</td>
<td>Lower bound on Y axis of matrix set-up debug output (switch 6).</td>
</tr>
<tr>
<td>Item 48</td>
<td>Upper bound on Y axis of matrix set-up debug output (switch 6).</td>
</tr>
<tr>
<td>Item 49</td>
<td>Lower bound on Z axis of matrix set-up debug output (switch 6).</td>
</tr>
<tr>
<td>Item 50</td>
<td>Upper bound on Z axis of matrix set-up debug output (switch 6).</td>
</tr>
<tr>
<td>Item 51</td>
<td>Debug from the History Matching option.</td>
</tr>
<tr>
<td>Item 52</td>
<td>Writes the RFT file as a GRAF user file, provided the output files are formatted.</td>
</tr>
<tr>
<td>Item 53</td>
<td>Debug from the flow reversal code.</td>
</tr>
<tr>
<td>Item 54</td>
<td>Fraction of the Newton direction used if <code>DEBUG</code> switch 38 is set.</td>
</tr>
<tr>
<td>Item 55</td>
<td>Debug from viscous displacement.</td>
</tr>
<tr>
<td>Item 56</td>
<td>Replaces bands by NNCs in the solver.</td>
</tr>
<tr>
<td>Item 57</td>
<td>Reserved for experimental features.</td>
</tr>
<tr>
<td>Item 58</td>
<td>Reserved for experimental features.</td>
</tr>
<tr>
<td>Item 59</td>
<td>Debug from timestep selection.</td>
</tr>
<tr>
<td>Item 60</td>
<td>Nested LGRs.</td>
</tr>
<tr>
<td>Item 61</td>
<td>Debug from petro-elastic model. The cells for which output is to be written are specified by switches 45-50.</td>
</tr>
<tr>
<td>Item 76</td>
<td>Debug from rivers. If set&gt;0 then a table of the river solution is output at every well iteration. If set&gt;1, solution obtained from river pre-solver is output.</td>
</tr>
<tr>
<td>Item 77</td>
<td>Debug from pseudo steady-state calculation.</td>
</tr>
</tbody>
</table>
### DEBUG item

<table>
<thead>
<tr>
<th>DEBUG item</th>
<th>Effect of DEBUG item</th>
</tr>
</thead>
<tbody>
<tr>
<td>Item 78</td>
<td>Statistics from SAMG solver setup and cycling.</td>
</tr>
<tr>
<td>Item 79</td>
<td>Outputs the norms of the residuals.</td>
</tr>
<tr>
<td>Item 80</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>Item 81</td>
<td>Reserved for experimental features.</td>
</tr>
<tr>
<td>Item 82</td>
<td>Debug from timestep selection.</td>
</tr>
<tr>
<td>Item 83</td>
<td></td>
</tr>
<tr>
<td>Item 84</td>
<td></td>
</tr>
<tr>
<td>Item 85</td>
<td></td>
</tr>
<tr>
<td>Item 86</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>Item 87</td>
<td>Reserved for developer use.</td>
</tr>
</tbody>
</table>

*Table 3.12:DEBUG keyword values*

#### Example

```
DEBUG 1 1 0 0 0 0 1 1 0 0 0 6*0 1 1 /
DEBUG 78* 1 /
```
Controls on debug output

The DEBUG3 keyword is primarily intended for use in development work, and may change from version to version. Most controls are only partially documented if at all, in particular for limitations and warnings on their use. Debug output is not fully supported.

In parallel, a DEBUG3 keyword, possibly empty, must be present in the RUNSPEC section for the debug output files to be created properly for each task. The run may fail otherwise.

Many of the items produce enormous amounts of debug output, and should be used with extreme caution.

The keyword should be followed by integers, each of which controls a form of debug output. The default value is zero, and greater values activate the appropriate output. The data field should be terminated by a slash (/). If fewer than the maximum number integers are read, the remainder are left unchanged. Once a particular debug item is activated, it remains activated in all subsequent uses of DEBUG3 keyword unless it is explicitly turned off.

<table>
<thead>
<tr>
<th>Item</th>
<th>Effect of DEBUG3 item</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Debug from allocation/keyword reading.</td>
</tr>
<tr>
<td></td>
<td>If 1, summary of storage allocation written to debug file.</td>
</tr>
<tr>
<td></td>
<td>If &gt;1, summary is also written to Print file, debug from allocation routines to Debug file.</td>
</tr>
<tr>
<td>2</td>
<td>Debug from file opening and closing.</td>
</tr>
<tr>
<td>3</td>
<td>Debug from table reading and checking.</td>
</tr>
<tr>
<td>4</td>
<td>Debug from equilibration.</td>
</tr>
<tr>
<td></td>
<td>Can be set to 1 or 2.</td>
</tr>
<tr>
<td>5</td>
<td>Debug from timestepping.</td>
</tr>
<tr>
<td>6</td>
<td>Debug from residual calculation.</td>
</tr>
<tr>
<td>7</td>
<td>Matrix evaluation by differencing.</td>
</tr>
<tr>
<td>8</td>
<td>Debug from non-linear solver.</td>
</tr>
<tr>
<td>9</td>
<td>Debug from linear solver.</td>
</tr>
<tr>
<td></td>
<td>Can be set to 1 or 2.</td>
</tr>
<tr>
<td>10</td>
<td>Clocks.</td>
</tr>
<tr>
<td></td>
<td>Can be set to 1 or 2. 2 is for ECLIPSE 100-style output.</td>
</tr>
<tr>
<td>11</td>
<td>Debug from well calculations.</td>
</tr>
<tr>
<td></td>
<td>Can be set to 1 or 2.</td>
</tr>
<tr>
<td>12</td>
<td>Debug from timestepping.</td>
</tr>
<tr>
<td>13</td>
<td>Debug of vector runs.</td>
</tr>
<tr>
<td>14</td>
<td>Debug from separators.</td>
</tr>
<tr>
<td>15</td>
<td>Debug of solution and properties.</td>
</tr>
<tr>
<td></td>
<td>Can be set to 1 or 2.</td>
</tr>
<tr>
<td>16</td>
<td>Debug from flash calculations.</td>
</tr>
<tr>
<td>17</td>
<td>Null check of analytical and numerical flow Jacobians.</td>
</tr>
<tr>
<td>Item</td>
<td>Effect of DEBUG3 item</td>
</tr>
<tr>
<td>------</td>
<td>-----------------------</td>
</tr>
<tr>
<td>18</td>
<td>Set monitor flag to on (-1 sets back to off).&lt;br&gt;  If set to 2, will flag all subroutine entry and exit points and write message to the debug flag.&lt;br&gt;  If set to 3, will also write message to the screen or the log file.</td>
</tr>
<tr>
<td>19</td>
<td>If set to 1, check inter-array gaps at each memory release.&lt;br&gt;  If set to 2, also check inter-array gaps at each subroutine entry and exit point.</td>
</tr>
<tr>
<td>20</td>
<td>Force divide check on fatal error to get trace back.</td>
</tr>
<tr>
<td>21</td>
<td>ACTION debug.</td>
</tr>
<tr>
<td>22</td>
<td>Basic vector flash debug:&lt;br&gt;  If set to 1: Summary debug (includes iteration counts, cell states and max residual).&lt;br&gt;  If set to 2: also input/output debug for individual cells.&lt;br&gt;  If set to 3: also output of residuals, C/D matrices, Z-factors + derivatives.</td>
</tr>
<tr>
<td>23</td>
<td>Output of matrix.</td>
</tr>
<tr>
<td>24</td>
<td>Debug from NNC calculations.&lt;br&gt;  Can be set to 1 or 2.</td>
</tr>
<tr>
<td>25</td>
<td>Debug from Michelsen vector stability test.&lt;br&gt;  If set to 1: Individual cell information from SS/BFGS algorithms.</td>
</tr>
<tr>
<td>26</td>
<td>Not used.</td>
</tr>
<tr>
<td>27</td>
<td>Newton Search direction debug facility (use with IDEBUG (40)).&lt;br&gt;  Turned on after NEWTIT &gt; IDEBUG (27), if IDEBUG (27)&gt;0.</td>
</tr>
<tr>
<td>28</td>
<td>Debug of B matrix</td>
</tr>
<tr>
<td>29</td>
<td>Newton Search direction debug facility (see DEBUG (27)).&lt;br&gt;  Specifies the number of sub-steps (up to a max. of 100).</td>
</tr>
<tr>
<td>30</td>
<td>Lower x-index for debug output.&lt;br&gt;  DEFAULT: 1</td>
</tr>
<tr>
<td>31</td>
<td>Upper x-index for debug output.&lt;br&gt;  DEFAULT: $N_x$</td>
</tr>
<tr>
<td>32</td>
<td>Lower y-index for debug output.&lt;br&gt;  DEFAULT: 1</td>
</tr>
<tr>
<td>33</td>
<td>Upper y-index for debug output.&lt;br&gt;  DEFAULT: $N_y$</td>
</tr>
<tr>
<td>34</td>
<td>Lower z-index for debug output.&lt;br&gt;  DEFAULT: 1</td>
</tr>
<tr>
<td>Item</td>
<td>Effect of DEBUG3 item</td>
</tr>
<tr>
<td>------</td>
<td>-----------------------</td>
</tr>
</tbody>
</table>
| 35   | Upper z-index for debug output.  
      | DEFAULT: $N_z$          |
| 36   | Lower active index for debug output.  
      | DEFAULT: 1              |
| 37   | Upper active index for debug output.  
      | DEFAULT: inactive       |
| 38   | Fuel calculation debug enabled.      |
| 39   | Debug of tracers.                |
| 40   | Solution and property debug for the cells defined by items 30-35. |
| 41   | Residual, solution and property debug. |
| 42   | Debug from SUMMARY section.        |
| 43   | State transition debug.            |
| 44   | Check linear solution by back-substitution. |
| 45   | Difference test of domain results.  |
| 46   | Aquifer debug.                     |
| 47   | Prevent pre-elimination of last equation/variable in implicit cases. |
| 48   | Select inner internal direction.   
      | Can be set to 1,2 or 3. (See also SOLVDIRS.) |
| 49   | Select outer internal direction.   
      | Can be set to 1,2 or 3. (See also SOLVDIRS.) |
| 50   | Debug of nested factorization pointers. |
| 51   | Debug of LGR.                     |
| 52   | PVTO and PVTG lookup debug.        |
| 53   | Hysteresis debug.                  |
| 54   | Gas plant table handling debug.    |
| 55   | VFP table handling debug.          |
| 56   | Select solution variable for domain debug.  
      | DEFAULT: all              |
| 57   | Null check by differencing of thermal flash:  
      | If set to 1: check flash Jacobian  
      | If set to 2: check property derivatives with respect to primaries, all states  
      | If set to 5-12: 2nd order difference of individual states  
<pre><code>  | OGW, OW, GW, W, OG, O, G, R, where 5=OGW, 6=OW and so on. |
</code></pre>
<p>| 58   | Miscible option debug.             |</p>
<table>
<thead>
<tr>
<th>Item</th>
<th>Effect of DEBUG3 item</th>
</tr>
</thead>
<tbody>
<tr>
<td>59</td>
<td>Non-equilibrium restart debug.</td>
</tr>
<tr>
<td>60</td>
<td>Debug from K-value generation.</td>
</tr>
<tr>
<td>61</td>
<td>Obtain extra properties for debug (specify before RUNSPEC).</td>
</tr>
<tr>
<td>62</td>
<td>Suppress space saving in matrix storage.</td>
</tr>
<tr>
<td>63</td>
<td>Force corner point transmissibility calculations.</td>
</tr>
</tbody>
</table>
| 64   | Debug from OVERTR, OVERTTHT, OVERTX, OVERTY, OVERTZ / TRANR, TRANHT, TRANX, TRANY, TRANZ keywords.  
If set to 2, report modified transmissibilities. |
| 65   | Not used. |
| 66   | Difference check of separator flash. |
| 67   | Suppress oil/gas relative permeability interpolation. |
| 68   | Debug from thermal flash. |
| 69   | Zudkevitch-Joffe debug. |
| 70   | Debug from thermal flow. |
| 71   | Force initial is at ion of all new allocated arrays.  
If set to 1 then initializes to zero.  
If set to 2 then initializes to an undefined value. |
| 72   | Not used. |
| 73   | Suppress NNC generation. |
| 74   | Debug of multisegment well and well derivatives:  
1. Check of Gamma inverse for each column J in LUD  
2. Check of solution of multisegment well matrix equation by multiplying out  
3. Check of full Jacobian by differencing  
4. Statistics of variable update (including well convergence information)  
5. Check of DMXP, derivatives of M-S well primary variables with respect to XP  
6. Check domain and MPRP inter-array gaps  
7. Check derivative of WRES with respect to BHP  
8. Check derivative of WRES with respect to XP (brief)  
9. Check derivative of WRES with respect to XP (verbose)  
10. Check derivative of Q with respect to XP (brief)  
11. Check derivative of Q with respect to XP (verbose). |
<p>| 75   | Debug from flux option. |</p>
<table>
<thead>
<tr>
<th>Item</th>
<th>Effect of DEBUG3 item</th>
</tr>
</thead>
</table>
| 76   | $P_{\text{sat}}$ debug.  
|      | 1: Trap $P_{\text{sat}}$ solution failures.  
|      | 2: $P_{\text{sat}}$ solution debug.  |
| 77   | Debug of heat loss to analytic rocks. Used in Thermal.  |
| 78   | Debug of thermal cutback.  |
| 79   | All completions cause AIM implicit cells, not just open ones.  |
| 80   | Parallel code debug.  |
| 81   | Force allocation of flow matrix.  |
| 82   | Not used.  |
| 83   | Force use of sudation NNC Jacobian generator.  |
| 84   | Not used.  |
| 85   | Velocity Dependent Relative Permeabilities.  
|      | If set to 1: Capillary Number Modification output.  
|      | If set to 2: Forchheimer Modification output.  
|      | If set to 3: Both outputs.  |
| 86   | Output Pseudo-Pressure Tables.  |
| 87   | Debug from vector flash (phase-split calculation).  
|      | If set to 1: Convergence of individual cells in GFE minimization.  
|      | If set to 2: Additional detail.  |
| 88   | Replace bands with NNCs.  
|      | If set to 1: replace them for outer directions.  
|      | If set to 2: replace them for all directions.  
<p>|      | If set to -1: suppress removal of inline NNCs.  |
| 89   | Request debug from the KZP routine (generates kzp*.dbg).  |
| 90   | Enable PSYNC to make more checks (note: this sets QMONTR=TRUE).  |
| 91   | If &gt; 0, skips Young re-ordering after an Appleyard chop.  |
| 92   | Debug <strong>WARP</strong> setup.  |
| 93   | Debug solution/wells output at each Newton iteration (<strong>ECLIPSE 100 DEBUG</strong> (19)).  |
| 94   | Rowsum debug $v = (B^* -1)Av$.  |
| 95   | Reserved.  |
| 96   | <strong>WARP</strong> LGR debug: if set to &gt;0 then sets up solver pointers for LGRs, blocks NNCs by to-from LGRs and sets QQWARP to true. If set to &gt;1, the NNCs are also blocked by <strong>AIM</strong> state within each group.  |
| 97   | <strong>GPMAINT</strong> debug.  |</p>
<table>
<thead>
<tr>
<th>Item</th>
<th>Effect of DEBUG3 item</th>
</tr>
</thead>
<tbody>
<tr>
<td>98</td>
<td>Override timestepping defaults:</td>
</tr>
<tr>
<td></td>
<td>1. IMPES</td>
</tr>
<tr>
<td></td>
<td>2. AIM</td>
</tr>
<tr>
<td></td>
<td>3. FULLIMP</td>
</tr>
<tr>
<td>99</td>
<td>Check on behavior of AIM.</td>
</tr>
<tr>
<td>100</td>
<td>Limited general debug.</td>
</tr>
<tr>
<td>101</td>
<td>Debug of multipoint flux &amp; tensor permeabilities.</td>
</tr>
<tr>
<td>102</td>
<td>Debug tests via Comsim test driver.</td>
</tr>
<tr>
<td>103</td>
<td>Debug of GEOMECH:</td>
</tr>
<tr>
<td></td>
<td>1. check full Jacobian - can be used with internal limits (value of item 30 to 35 or 36 to 37)</td>
</tr>
<tr>
<td></td>
<td>2. write out matrix in TMCRWS format</td>
</tr>
<tr>
<td></td>
<td>3. not used</td>
</tr>
<tr>
<td></td>
<td>4. solve matrix with direct LU decomposition solver</td>
</tr>
<tr>
<td></td>
<td>5. only perform direct solving if WARP or ECLSOl fails, then only compare answers</td>
</tr>
<tr>
<td></td>
<td>6. same as 1, but only check geomech variables</td>
</tr>
<tr>
<td></td>
<td>7. check that the plastic yield function is being zeroed</td>
</tr>
<tr>
<td></td>
<td>8. check derivatives numerically of yield function with respect to stress</td>
</tr>
<tr>
<td></td>
<td>9. statistics on plastic calculation</td>
</tr>
<tr>
<td></td>
<td>10. if moving grid, delay copying latest pore volume until XGMGUD</td>
</tr>
<tr>
<td></td>
<td>11. Plastic iteration summary</td>
</tr>
<tr>
<td></td>
<td>12. SAMG solver</td>
</tr>
<tr>
<td></td>
<td>13. Reserved for developer use</td>
</tr>
<tr>
<td></td>
<td>14. FAULT NNC summary.</td>
</tr>
<tr>
<td>104</td>
<td>Experimental treatment of well terms for the linear solver.</td>
</tr>
<tr>
<td>105</td>
<td>Improved black oil formulation:</td>
</tr>
<tr>
<td></td>
<td>If set to 1: Non-linear convergence</td>
</tr>
<tr>
<td></td>
<td>If set to 2: +linear convergence.</td>
</tr>
<tr>
<td>106</td>
<td>Debug from network simulator and balancing calculations. (Network Option.)</td>
</tr>
<tr>
<td>107</td>
<td>Velocities and radii at pressure points in pseudo calculation (compared with velocity-dependent $K_r$).</td>
</tr>
<tr>
<td>108</td>
<td>Initialize derivatives in domain calculations.</td>
</tr>
<tr>
<td></td>
<td>If set to 1: initialized to 0.</td>
</tr>
<tr>
<td></td>
<td>If set to 2: initialized to NaN.</td>
</tr>
<tr>
<td>Item</td>
<td>Effect of DEBUG3 item</td>
</tr>
<tr>
<td>------</td>
<td>-----------------------</td>
</tr>
<tr>
<td>109</td>
<td>Compares the results from skipping a stability calculation with a rigorous stability calculation. (Fast Flash calculation.)</td>
</tr>
<tr>
<td>110</td>
<td>Debug from the Gradient Option.</td>
</tr>
<tr>
<td>111</td>
<td>Debug from PEBI grids (global/local/solver mapping).</td>
</tr>
<tr>
<td>112</td>
<td>For stability debug.</td>
</tr>
<tr>
<td>113</td>
<td>Reserved.</td>
</tr>
<tr>
<td>114</td>
<td>Reserved.</td>
</tr>
<tr>
<td>115</td>
<td>Debug of group solving.</td>
</tr>
<tr>
<td>116</td>
<td>Detailed well debug.</td>
</tr>
<tr>
<td>117</td>
<td>Debug from conductive faults.</td>
</tr>
<tr>
<td>118</td>
<td>Debug for FrontSim material balance errors.</td>
</tr>
<tr>
<td>119</td>
<td>Debug sector model.</td>
</tr>
<tr>
<td>120</td>
<td>Production optimization debug:</td>
</tr>
<tr>
<td>121</td>
<td>Adjoint gradient debug:</td>
</tr>
<tr>
<td>122</td>
<td>Detailed (non-linear) convergence debug.</td>
</tr>
<tr>
<td>123</td>
<td>Derivative checker for conductive faults:</td>
</tr>
<tr>
<td>124</td>
<td>Reserved.</td>
</tr>
<tr>
<td>125</td>
<td>Perturbation in numerical gradient computation is $10^{-(\text{value of item 125})}$.</td>
</tr>
</tbody>
</table>

**Keywords**

582
<table>
<thead>
<tr>
<th>Item</th>
<th>Effect of DEBUG3 item</th>
</tr>
</thead>
<tbody>
<tr>
<td>126</td>
<td>Adjoint partial gradient calculation options:</td>
</tr>
<tr>
<td></td>
<td>Bit-packed:</td>
</tr>
<tr>
<td></td>
<td>1 = analytic dFdP (only implemented for certain parameters)</td>
</tr>
<tr>
<td></td>
<td>2 = analytic dRdP, (only implemented for certain parameters)</td>
</tr>
<tr>
<td></td>
<td>4 = numerical dFdX. (not advised: output is extremely verbose).</td>
</tr>
<tr>
<td>127</td>
<td>Reserved.</td>
</tr>
<tr>
<td>128</td>
<td>Reserved.</td>
</tr>
<tr>
<td>129</td>
<td>If set to 1: dump prior to flash.</td>
</tr>
<tr>
<td></td>
<td>If set to 2: dump after flash.</td>
</tr>
<tr>
<td></td>
<td>If set to 3: dump prior to and after flash.</td>
</tr>
<tr>
<td>130</td>
<td>Time linear solver timer:</td>
</tr>
<tr>
<td></td>
<td>iterations = mod(100, item 130 value)</td>
</tr>
<tr>
<td></td>
<td>solver iterations = mod(10, item 130 value).</td>
</tr>
<tr>
<td>131</td>
<td>AIM done per color = 1.</td>
</tr>
<tr>
<td>132</td>
<td>Turn off WARP NNCs.</td>
</tr>
<tr>
<td>133</td>
<td>WARP preconditioner.</td>
</tr>
<tr>
<td>134</td>
<td>If set to 2 and FULLIMP then do NOT break up IVRB runs.</td>
</tr>
<tr>
<td>135</td>
<td>Reserved.</td>
</tr>
<tr>
<td>136</td>
<td>Heater derivative check.</td>
</tr>
<tr>
<td>137</td>
<td>Reserved.</td>
</tr>
<tr>
<td>138</td>
<td>ILU0 solver debug.</td>
</tr>
<tr>
<td>139</td>
<td>Reserved.</td>
</tr>
<tr>
<td>140</td>
<td>Reserved.</td>
</tr>
<tr>
<td>141</td>
<td>Linear system in sparse (compressed row) format</td>
</tr>
<tr>
<td></td>
<td>(NEWTIT = (value of item 141)-1)</td>
</tr>
<tr>
<td>142</td>
<td>DEBUG3 items for the Gas Lift Optimization calculation are:</td>
</tr>
<tr>
<td></td>
<td>Set to 1 for basic debug</td>
</tr>
<tr>
<td></td>
<td>Set to 2 for above and well gas lift increment/decrement debug</td>
</tr>
<tr>
<td></td>
<td>Set to 3 for above and dead well debug</td>
</tr>
<tr>
<td></td>
<td>Set to 4 for above and estimated group flow rates debug</td>
</tr>
<tr>
<td></td>
<td>Set to 6 for above and well proxy model debug.</td>
</tr>
<tr>
<td>143</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>144</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>145</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>Item</td>
<td>Effect of DEBUG3 item</td>
</tr>
<tr>
<td>------</td>
<td>-----------------------</td>
</tr>
<tr>
<td>146</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>147</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>148</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>149</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>150</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>151</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>152</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>153</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>154</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>155</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>156</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>157</td>
<td>If &gt; 0 outputs extra arrays for the geomechanics data structure. Only useful for developers.</td>
</tr>
<tr>
<td>158</td>
<td>Reserved for developer use.</td>
</tr>
<tr>
<td>159</td>
<td>Reserved for developer use.</td>
</tr>
</tbody>
</table>
| 160  | Used by the CO2STORE option  
Set to 1 output when the liquid salt concentrations are high. For NACL this is the point of crystallization. For CACL2 this is for concentrations above 4 Molal. |
| 161  | Reserved for developer use. |
| 162  | Reserved for developer use. |
| 163  | Reserved for developer use. |
| 164  | Reserved for developer use. |
| 165  | Reserved for developer use. |
| 166  | Activated when set to an integer greater than 0.  
Output the largest phase relative permeability derivatives together with the active index of the worst-offending cell.  
If the largest derivative is unique to a particular cell, then this cell is highlighted.  
If two or more cells have the largest derivative, then just one of these cells will be highlighted.  
Compatible with all options. |
| 167  | Used by the GEOCHEM option.  
Activated when set to 1.  
Outputs details of cells that did not converge during geochemistry flash computations. |
| 168  | Reserved. |
| 169  | Reserved for developer use. |

*Table 3.13: DEBUG3 keyword values*
Example

```plaintext
DEBUG3
4*0 1 1 0 1 /
DEBUG3
120* 1 /
```


## DELAYACT

**Specifies a set of keywords to be processed after an ACTION is triggered**

The `DELAYACT` keyword marks the start of a set of `SCHEDULE` section keywords that are to be stored for later processing when a nominated `ACTION` has been triggered and a specified amount of time has passed. The set of keywords must be terminated with the `ENDACTIO` keyword. The keywords between the `DELAYACT` and `ENDACTIO` keywords are processed at the end of the timestep when the condition defined in the `DELAYACT` keyword is satisfied. You can choose whether the action is to be performed only once, or repeatedly each timestep while the triggering condition is satisfied.

In ECLIPSE 100, almost any `SCHEDULE` section keyword may be placed between `DELAYACT` and `ENDACTIO`, with the exception of timestepping keywords (for example `TIME`, `TSTEP`, `DATES` and `GASYEAR`), the global keywords, and the `INCLUDE` keyword.

In ECLIPSE 300, practically any `SCHEDULE` section keyword may be placed between `DELAYACT` and `ENDACTIO`, with the exception of keywords described for `ACTIONACTIONG`, `ACTIONR`, `ACTIONW`, `ACTIONS`, `ACTIONX`).

The amount of storage space reserved for these keywords is controlled by the `RUNSPEC` section keyword `ACTDIMS`. The number of lines between a `DELAYACT` keyword and its corresponding `ENDACTIO` keyword (including the `ENDACTIO` keyword itself) is limited to `MNLPAC` in `ACTDIMS` item 2. The length of each line is limited to `NCPLIN` in `ACTDIMS` item 3.

Any number of action keyword sets may be declared, each with its own triggering condition; the maximum number being limited to `MAXACT` in `ACTDIMS` item 1. Each action keyword set must be bracketed by a pair of `ACTION`, `ACTIONG`, `ACTIONR`, `ACTIONW`, `ACTIONS`, `ACTIONX` and `ENDACTIO` keywords, and is distinguished by its action name defined in item 1. This name must be unique across all the action keyword family. If an `ACTION`, `ACTIONG`, `ACTIONR`, `ACTIONW`, `ACTIONS`, `ACTIONX` keyword is re-entered with the same action name as a previously declared action, the previously declared action is deleted and replaced with the new action.

The `DELAYACT` keyword is followed by the following items of data, terminated with a slash (/):

1. **The action name.**
   - A name of up to 8 characters, to distinguish this action from any others.
   - If an action with this name has already been declared in any of the action keyword family `ACTION`, `ACTIONG`, `ACTIONR`, `ACTIONW`, `ACTIONS`, `ACTIONX` and `DELAYACT` it is overwritten with the action data specified below.

2. **Action name that triggers this action**

3. **The delay after which this action will be triggered, once the action in item 2 has been triggered. Must be greater than zero.**
   - **UNITS:** day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

4. **Number of times this action can be triggered.**
   - The action is performed once at the end of each timestep while the action named in item 2 satisfies the triggering condition, until it has been performed the specified number of times. A number of 10,000 or greater is interpreted as ‘infinity’.
   - **DEFAULT:** 1

5. **Increment to the triggering condition.**

---

### Keywords

- `ECLIPSE 100`
- `ECLIPSE 300`
- `SPECIAL`
- `RUNSPEC`
- `GRID`
- `EDIT`
- `PROPS`
- `REGIONS`
- `SUMMARY`
- `SCHEDULE`
After each time the action is performed, the specified increment is added to the triggering condition value (initialized in item 3). The increment may be negative, if desired.

**UNITS**: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

**DEFAULT**: 0.0

End the data record with a slash (/).

**Note**: If the triggering action gets triggered repeatedly (in particular, before this action has been eliminated) the clock for triggering this action is not reset. That is, the countdown for the triggering of this action commences from the first time the action in item 2 is triggered.

**CAUTION**: It is possible to nest actions by including the definition of the ‘inner’ action within the keyword set of the ‘outer’ action (that is, before its **ENDACTIO** keyword). This offers great flexibility, but care should be exercised and the consequences of the nested actions should be thought through. For example: suppose that in the **ACTDIMS** keyword we have set **MAXACT=1** and have then defined an action **ACT1**. Suppose also that as part of this action we have defined another action, **ACT2**. **ECLIPSE** will only issue an error when **ACT1** is actually implemented, which may, of course, be well into the run. Alternatively we may call the inner action by the same name as the outer action, **ACT1**. This is perfectly allowable, but it must be remembered that each time an action of a given name is defined it overwrites the properties that the action may previously have had.

Suppose that we have an outer action that can be repeated a number of times. Within this action is an inner one that can also be repeated and that also increments its triggering condition each time. No matter how many times the inner action has been called, when the outer action is invoked again the inner action is redefined to its original specification.

**Note**: Full checking of the keywords entered with an action keyword cannot be performed until the action is actually implemented, since it is possible to use wild cards in many of the keywords. However, before a simulation run is attempted the data can be run in **NOSIM** mode. This mode does whatever checks it can on the keywords within any action keyword blocks that have been defined. All the most common types of errors of syntax and typing will be found, but errors resulting from wild cards feeding inappropriate well/group names to keywords are not found.

**Note**: As with the other **ACTION** type keywords, **DELAYACT** does not affect the report times or the overall length of a run. This is still fully determined by the **TIME**, **TSTEP** and **DATES** keywords following any **ACTION** type keywords. A run is not extended beyond the time it is to finish based on **TIME**, **TSTEP** and **DATES** information just because there is a delayed action waiting to be performed.

**Example**

This example shows the **DELAYACT** keyword being used to help control a water reservoir. An **ACTIONW** keyword is used to close the producing well when the brine concentration becomes unacceptable. Water injection then takes place. At the same time a **DELAYACT** action is defined and this essentially allows the injection to go ahead for 210 days before the injection is turned off and production restarts. The cycle can then be repeated. Here we have chosen to shut all the wells the next time the brine concentration limit is violated, and a **DELAYACT** keyword has been used just to let the run continue for another 20 days so that graphics of the results will not appear to end quite so suddenly.
ACTIONW
'STOPPROD' 'PRODUCER' WSPC > 0.25 /
COMPDAT
'INJECTOR' 1 1 1 10 'OPEN' 0 1* .6096 /
'INJECTOR' 1 2 1 10 'OPEN' 0 1* .6096 /
'INJECTOR' 1 3 1 10 'OPEN' 0 1* .6096 /
'INJECTOR' 1 4 1 10 'OPEN' 0 1* .6096 /
'PRODUCER' 1 1 1 10 'SHUT' 0 1* .6096 /
'PRODUCER' 1 2 1 10 'SHUT' 0 1* .6096 /
'PRODUCER' 1 3 1 10 'SHUT' 0 1* .6096 /
'PRODUCER' 1 4 1 10 'SHUT' 0 1* .6096 /
/
-- PRODUCTION WELL CONTROLS
--
-- WELL OPEN/ CNTL OIL WATER GAS LIQU RES BHP
-- NAME SHUT MODE RATE RATE RATE RATE RATE
WCONINJE
'INJECTOR' 'WATER' 'OPEN' 'RATE' 68192. /
/
WSALT
'INJECTOR' 0 /
/
WCONPROD
'PRODUCER' 'SHUT' 'WRAT' 1* 0. /
/
DELAYACT
'STARTPRO' 'STOPPROD' 210 /
COMPDAT
'INJECTOR' 1 1 1 10 'SHUT' 0 1* .6096 /
'INJECTOR' 1 2 1 10 'SHUT' 0 1* .6096 /
'INJECTOR' 1 3 1 10 'SHUT' 0 1* .6096 /
'INJECTOR' 1 4 1 10 'SHUT' 0 1* .6096 /
'PRODUCER' 1 1 1 10 'OPEN' 0 1* .6096 /
'PRODUCER' 1 2 1 10 'OPEN' 0 1* .6096 /
'PRODUCER' 1 3 1 10 'OPEN' 0 1* .6096 /
'PRODUCER' 1 4 1 10 'OPEN' 0 1* .6096 /
/
-- PRODUCTION WELL CONTROLS
--
-- WELL OPEN/ CNTL OIL WATER GAS LIQU RES BHP
-- NAME SHUT MODE RATE RATE RATE RATE RATE
WCONINJE
'INJECTOR' 'WATER' 'SHUT' 'RATE' 0. /
/
WCONPROD
'PRODUCER' 'OPEN' 'WRAT' 1* 68192. /
/
ACTIONW
'ENDRUN' 'PRODUCER' WSPC > 0.25 /
COMPDAT
'INJECTOR' 1 1 1 10 'SHUT' 0 1* .6096 /
'INJECTOR' 1 2 1 10 'SHUT' 0 1* .6096 /
'INJECTOR' 1 3 1 10 'SHUT' 0 1* .6096 /
'INJECTOR' 1 4 1 10 'SHUT' 0 1* .6096 /
'PRODUCER' 1 1 1 10 'OPEN' 0 1* .6096 /
'PRODUCER' 1 2 1 10 'OPEN' 0 1* .6096 /
'PRODUCER' 1 3 1 10 'OPEN' 0 1* .6096 /
'PRODUCER' 1 4 1 10 'OPEN' 0 1* .6096 /
/
ECLIPSE Reference Manual

WCONPROD
   'PRODUCER' 'SHUT' 'WRAT' 1* 0. / /

DELAYACT
   'ENDRUN20' 'ENDRUN' 20 /

END
ENDACTIO
ENDACTIO
ENDACTIO
ENDACTIO

Keywords

589
Specify coefficients of Ezrokhi’s method for aqua density

This keyword enables the default Ezrokhi coefficients for the aqua density calculation to be overwritten. The data consists of a table composed of a series of 3 coefficients for each component which may dissolve in the aqua phase. In GASSOL, these components are specified using the SOLUAQA table; within CO2STORE, GASWAT and thermal with KWTABTn. Ezrokhi’s coefficients are specified for all components, although it should be noted that the coefficients for component H2O are automatically taken to be zero, irrespective of alternative user-supplied inputs. This keyword associates a contribution of each dissolved component into the aqua density calculation. See also “GASSOL Solution in the Aqueous Phase” in the ECLIPSE Technical Description.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

This keyword is thus compatible with the CO2STORE, GASWAT, GASSOL and KWTABTn (THERMAL) options.

Examples

Example 1

Input data for a four-component study:

```
DENAQA
2.01E-6 -6.34E-7 1.31E-9
1.01E-6 -5.34E-7 2.31E-9
2.01E-6 -4.34E-7 3.31E-9
2.03E-6 -4.54E-7 3.61E-9
/
```

Example 2

With two reservoir EoS regions and four-component study:

```
DENAQA
2.01E-6 -6.34E-7 1.31E-9
1.01E-6 -5.34E-7 2.31E-9
2.01E-6 -4.34E-7 3.31E-9
2.03E-6 -4.54E-7 3.61E-9
/
2.02E-6 -6.24E-7 1.21E-9
1.02E-6 -5.24E-7 2.21E-9
2.02E-6 -4.24E-7 3.21E-9
2.02E-6 -4.24E-7 3.21E-9
/```
## DENSITY

### Fluid densities at surface conditions

The data comprises three fluid densities for each pressure table region. The number of pressure table regions is specified using keyword `TABDIMS` in the `RUNSPEC` section and defaults to 1. Each record consists of the following items of data, and is terminated by a slash (/).

1. **The density of oil at surface conditions.**
   - UNITS: kg/m$^3$ (METRIC), lb/ft$^3$ (FIELD), gm/cc (LAB), kg/m$^3$ (PVT-M)
   - ECLIPSE 300 DEFAULT: 600 kg/m$^3 = 37.457$ lb/ft$^3$

2. **The density of water at surface conditions.**
   - UNITS: kg/m$^3$ (METRIC), lb/ft$^3$ (FIELD), gm/cc (LAB), kg/m$^3$ (PVT-M)
   - ECLIPSE 300 DEFAULT: 999.014 kg/m$^3 = 62.366$ lb/ft$^3$

3. **The density of gas at surface conditions.**
   - UNITS: kg/m$^3$ (METRIC), lb/ft$^3$ (FIELD), gm/cc (LAB), kg/m$^3$ (PVT-M)
   - ECLIPSE 300 DEFAULT: 1 kg/m$^3 = 0.062428$ lb/ft$^3$

   The keyword `GRAVITY` may be used as an alternative to `DENSITY`.

### Examples

#### Example 1

For a black oil case, all three phase densities are specified:

```
DENSITY 45.0000  63.0200   .07020 /
```

#### Example 2

For a compositional run, only the water density is required:

```
DENSITY 1* 62.0 1* /
```

---

**ECLIPSE Reference Manual**

**Keywords**

591
Grid block center depths

The keyword should be followed by one number for every grid block in the current input box, specifying the depth at the center of the grid block. The data must be terminated by a slash (/).

This keyword is entirely optional. Any depths that are not altered remain at the values calculated by ECLIPSE from the GRID data.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*6027). Note that spaces must not be inserted on either side of the asterisk.

Note: If the depth of a refined global cell is altered in the EDIT section, these changes are not applied to the local cells. Use REFINE to change the local cells as well.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

Example

```
----------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX         11 14  3  8  6  6   /
DEPTH 24*6094 /
```
DEPTHTAB Tables of river depth data

This keyword is used to define the downstream depth boundary condition of a river as a function of the time.

The keyword is followed by NMDEPT tables with a maximum of MXDEPT entries in each. These numbers are set with the RIVRDIMS keyword. Each table is terminated with a slash (/).

Each table consists of 5 columns of data:
1. Day
2. The name of the month abbreviated to three characters
   JAN, FEB, MAR, APR, MAY, JUN, JUL, AUG, SEP, OCT, NOV or DEC.
   JUL is an acceptable alternative to JLY.
3. Year
   Input the year as a four-digit integer number.
4. Time of observation in 24 hour clock format, hh:mm
   Do not use 24:00; use 00:00 of the following day
5. Depth at this time.
   UNITS: m (METRIC), ft (FIELD), cm (LAB)

The minimum number of rows of data per table is two. End the table with a slash (/). After the first tables subsequent tables may be defaulted to the previous table by just entering a slash.

The times, items 1-3 considered as a whole, must be monotonically increasing down the table.

Example

```
DEPTHTAB
  1 JAN 1995 00:00 1
  8 FEB 1995 2:45 2
 15 SEP 1995 12:00 3
/
```
DETAILMF  Initial mole fraction make-up of lumped components versus depth

This keyword specifies the depth tables to be used for initializing the concentration of the tracers representing detailed components in a lumped simulation (keyword LUMPDIMS in the RUNSPEC section). The data consists of NTTRVD tables (item 4 of the EQLDIMS keyword), with one table for each tracer region defined under keyword TNUM.

CAUTION: The order of the entries in the tables is very important and should match exactly the order in which the lumpings and detailed components were entered in the LUMPING keyword.

The data consists of a depth followed by the mole fraction for each detailed component for each lumping. Each table must be terminated with a slash (/).

Column 1  Depth values.
   The values should increase monotonically down the column.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

Column 2  The mole fraction of the first lumped component to be associated with the first detailed component of this lumping.

Column 3  The mole fraction of the first lumped component to be associated with the second detailed component of this lumping.

...  

...  

Column n  The mole fraction of the second lumped component to be associated with the first detailed component of this lumping.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSTRVD (see keyword EQLDIMS).

Example

For the example in the LUMPING keyword.

```
DETAILMF
  --Table 1
  1.0 0.05 0.15
   0.25 0.25 0.3
  1000.0 0.0 0.5
   0.25 0.25 0.0/
  --Table 2
  1.0 1.0 0.0
   0.0 0.5 0.0
  10000.0 0.0 0.0
   0.0 0.5 0.5/
  --Table 3
  1.0 1.0 0.0
  10000.0 0.9 0.07 0.03 /
```
**DETAILVD**

**Initial fraction make-up of lumped components versus depth**

This keyword specifies the depth tables to be used for initializing the concentration of the tracers representing detailed components in a lumped simulation (keyword `LUMPdims` in the RUNSPEC section). The data consists of `NTRVD` tables (item 4 of the `EQLDIMS` keyword), with one table for each tracer region defined under keyword `TNUM`.

**CAUTION:** The order of the entries in the tables is very important and should match exactly the order in which the lumpings and detailed components were entered in the `LUMPING` keyword.

The data consists of a depth followed by a fraction for each detailed component for each lumping. The fraction represents the proportion of the mole fraction of the carrier component to be associated with that tracer/detailed component. For each lumped phase the fractions must add up to 1.

Each table must be terminated with a slash (/)

Column 1  Depth values.

   The values should increase monotonically down the column.

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

Column 2  The fraction of the first lumped component to be associated with the first detailed component of this lumping.

Column 3  The fraction of the first lumped component to be associated with the second detailed component of the lumping.

Column n  The fraction of the second lumped component to be associated with the first detailed component of this lumping.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than `NSTRVD` (see keyword `EQLDIMS`).

**Example**

For the example in the `LUMPING` keyword.

```
DETAILVD
   --Table 1
   1.0  0.5  0.5
   0.25  0.25  0.5
   10000.0  0.5  0.5
   0.25  0.25  0.5/
   --Table 2
   1.0  0.0  1.0
   0.0  0.5  0.5
   10000.0  0.0  1.0
   0.0  0.5  0.5/
   --Table 3
```

---

**Keywords**
<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
<td>1.0</td>
<td>0.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10000.0</td>
<td>0.9</td>
<td>0.07</td>
<td>0.03</td>
<td>/</td>
</tr>
</tbody>
</table>
**DGRDT**

**Maximum rate of change of guide rate for wells under group control**

When wells use the default guide rate option (using potentials rather than user assigned values or an equation defined by `GUIDERAT` keyword) there are cases, particularly with highly productive horizontal wells, when the guide rates can vary excessively from step to step. This keyword may be used to constrain the current timestep’s guide rate to fall within bounds defined by the previous timestep’s guide rate.

The keyword is followed by a line containing some or all of the following items of data, terminated with a slash (/).

The items are:

1. Limiting fractional rate of change of guide rate.

   A lower fractional bound on the guide rate is formed by multiplying the value in this item by the current timestep and subtracting the result from 1.0.

   An upper fractional bound on the guide rate is formed by multiplying the value in this item by the current timestep and adding the result to 1.0.

   The limiting bounds on the current timestep’s guide rate are formed by multiplying the above fractional bounds by the previous timestep’s guide rate.

   **UNITS:** 1/day (METRIC), 1/day (FIELD), 1/hr (LAB), 1/day (PVT-M)

   **DEFAULT:** 100.0 (all unit sets)

2. The smallest permitted lower fractional bound as calculated above.

   **DEFAULT:** 0.5

3. The largest permitted upper fractional bound as calculated above.

   **DEFAULT:** 1.5

**Example**

Allow guide rate changes of 1% per day:

```plaintext
DGRDT 0.01 /
```

**Keywords**

597
**DGRID**

**Dump section of grid**

This keyword dumps a box of the current model to a .DGRD file. This file contains the geometric grid information, along with any input solution vectors for the defined box, in the GRID, EDIT, PROPS, REGIONS and SOLUTION sections of the current data set. The main purpose of this keyword is to use the .DGRD file to produce a smaller sector model from the current dataset. The keyword contains the following items:

1. First block on X axis of the box (IX1).
   
   DEFAULT: 1

2. Last block on X axis of the box (IX2).
   
   DEFAULT: NDIVIX

3. First block on Y axis of the box (JY1).
   
   DEFAULT: 1

4. Last block on Y axis of the box (JY2).
   
   DEFAULT: NDIVIY

5. First block on Z axis of the box (KZ1).
   
   DEFAULT: 1

6. Last block on Z axis of the box (KZ2).
   
   DEFAULT: NDIVIZ

7. LGR name. A blank is interpreted as the base grid.
   
   DEFAULT: ' '

The data must satisfy:

\[ 1 \leq IX1 \leq IX2 \leq NDIVIX \]

\[ 1 \leq JY1 \leq JY2 \leq NDIVIY \]

\[ 1 \leq KZ1 \leq KZ2 \leq NDIVIZ \]  

(where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS)

The data must be terminated by a slash (/).

If the dataset is a restart run (using the flexible restart facility (that is the RESTART keyword in the SOLUTION section), then the keywords SWAT, SOIL, PRESSURE, XMF and YMF will be output to the .DGRD file. This enables a non-equilibrium restart of the sector model.

If the current dataset is corner point, then the corner point keywords COORD and ZCORN are included in the GRID section of the .DGRD file. Similarly for block centered keywords. If a keyword in the current dataset is derived, for example, by using EQUALS, BOX and COPY keywords, this derivation is NOT reproduced in the .DGRD file.

Note only the keywords that are solution vectors, over NX, NY, NZ are output to the .DGRD file. For example the ACTNUM data from the GRID section or the FIPNUM data from the REGIONS section.
CAUTION: This keyword does not work with either LGRs or the PARALLEL keyword.

Example

```
GRID
-- output section 24-55, 128-140, 1-11 to the DGRD file
DGRID
24 55 128 140 1 11 /
```
Enables alternate form of tracer dispersion

There is no data associated with this keyword

**CAUTION:** Whilst, for simplicity, we use (x,y,z) below, these are not actually the coordinates used. The coordinate system used for dispersion follows the rock layers; that is we actually require data for, and do the calculations in, the i-direction, the j-direction and the k-direction. These may not be the same as (x,y,z) coordinates.

The form of dispersion used when this keyword is used replaces the dispersion matrix described in the TRDIS keyword with the one below:

\[
\begin{pmatrix}
A_1 & 0 & 0 \\
0 & A_2 & 0 \\
0 & 0 & A_3 \\
\end{pmatrix}
\]

where:

\[
A_1 = D_1(\mid v_x \mid \cdot c) + D_2(\mid v_y \mid \cdot c) + D_3(\mid v_z \mid \cdot c)
\]

\[
A_2 = D_4(\mid v_x \mid \cdot c) + D_5(\mid v_y \mid \cdot c) + D_6(\mid v_z \mid \cdot c)
\]

\[
A_3 = D_7(\mid v_x \mid \cdot c) + D_8(\mid v_y \mid \cdot c) + D_9(\mid v_z \mid \cdot c)
\]

Note that for a brine tracer, if the DSPDEINT keyword has been used, the concentrations in the lookup tables are replaced with the cell’s water density.

**Note:** The option does not work with two-phase or partitioned tracers.

See also keywords DISPDIMS, DISPERSE and DSPDEINT.

**Example**

DIAGDISP
**DIFFAGAS  Activity corrected gas diffusion coefficients**

This keyword specifies a gas phase activity-corrected diffusion coefficient for each component in a compositional run. These are used to define gas phase diffusive flows in terms of chemical potential differences.

The activity-corrected diffusion coefficient is defined in Reid, Prausnitz and Polling ([Ref. 90] in ECLIPSE Technical Description). These are also described in "Diffusion" in the ECLIPSE Technical Description. Activity-related coefficients may be related to the normal form of diffusion coefficient by:

\[ D_i^a = D_i \frac{\partial}{\partial \ln y_i} (\ln f_i) \]  
Eq. 3.52

where the normal diffusion coefficients are defined by the condition:

\[ J_i = -cD_i \frac{\partial y_i}{\partial d} \]  
Eq. 3.53

where

- \( c \) is the total molar concentration.
- \( J_i \) is the flux of component \( i \) per unit area.
- \( D_i \) is the diffusion coefficient of component \( i \).
- \( \frac{\partial y_i}{\partial d} \) is molar concentration gradient of component \( i \).

For gases, the normal and activity-corrected diffusion coefficients are similar, and become equal at low pressure.

The activity-corrected diffusion coefficients are used in ECLIPSE 300 to obtain inter block diffusive flows in terms of log fugacity (chemical potential) differences. The gas phase diffusive flow is:

\[ F_{ig}^{diff} = T_D D_i^a \left(y_i S_S h_i^m\right) \frac{\partial}{\partial \ln f_i} \left(M_i G h - \frac{M_i G h}{RT}\right) \]  
Eq. 3.54

where \( T_D \) is the diffusivity, the analogue of transmissibility for diffusive flow, and \( f_i \) are the component fugacities. For more details, see "Diffusion" in the ECLIPSE Technical Description. Diffusivities should be entered in the appropriate unit set. If the values available are in \( \text{cm}^2/\text{s} \), they may be converted as follows:

- **LAB units:** 1 \( \text{cm}^2/\text{s} = 3600 \text{ cm}^2/\text{hr} \)
- **METRIC or PVT–M units:** 1 \( \text{cm}^2/\text{s} = 8.64 \text{ m}^2/\text{day} \)
- **FIELD units:** 1 \( \text{cm}^2/\text{s} = 92.9979 \text{ ft}^2/\text{day} \)
- **UNITS:** \( \text{m}^2/\text{day} \) (METRIC), \( \text{ft}^2/\text{day} \) (FIELD), \( \text{cm}^2/\text{hr} \) (LAB), \( \text{m}^2/\text{day} \) (PVT–M)
**DIFFAOIL**

**Activity corrected oil diffusion coefficients**

This keyword specifies an oil phase activity-corrected diffusion coefficient for each component in a compositional run. These are used to define oil phase diffusive flows in terms of chemical potential differences. Oil diffusion coefficients are typically an order of magnitude lower than gas diffusion coefficients.

The units and format of the **DIFFAOIL** keyword are entirely analogous to the **DIFFAGAS** keyword. 

UNITS: $m^2$/day (METRIC), $ft^2$/day (FIELD), $cm^2$/hr (LAB), $m^2$/day (PVT-M)
Molecular diffusion data for each PVT region

This keyword is used to specify the fluid diffusion data for runs using the molecular diffusion option (keyword DIFFUSE in the RUNSPEC section). The data comprises NTPVT (see keyword TABDIMS) records of molecular weights and diffusion coefficients, one for each PVT region.

Each record consists of 8 data items terminated by a slash (/):

1. The oil molecular weight.
   
   UNITS: kg/kg-M (METRIC), lb/lb-M (FIELD), gm/gm-M (LAB)

2. The gas molecular weight.
   
   UNITS: kg/kg-M (METRIC), lb/lb-M (FIELD), gm/gm-M (LAB)

   
   UNITS: m²/day (METRIC), ft²/day (FIELD), cm²/hr (LAB)
   DEFAULT: Gas-in-gas coefficient’s value

   
   UNITS: m²/day (METRIC), ft²/day (FIELD), cm²/hr (LAB)
   DEFAULT: Gas-in-oil coefficient’s value

5. The gas-in-oil diffusion coefficient.
   
   UNITS: m²/day (METRIC), ft²/day (FIELD), cm²/hr (LAB)
   DEFAULT: Gas-in-oil coefficient’s value

6. The oil-in-oil diffusion coefficient.
   
   UNITS: m²/day (METRIC), ft²/day (FIELD), cm²/hr (LAB)
   DEFAULT: Gas-in-oil coefficient’s value

   
   UNITS: m²/day (METRIC), ft²/day (FIELD), cm²/hr (LAB)
   DEFAULT: 0.0

8. The oil-in-oil diffusion coefficient for cross phase diffusion.
   
   UNITS: m²/day (METRIC), ft²/day (FIELD), cm²/hr (LAB)
   DEFAULT: 0.0

ECLIPSE calculates the diffusive flow from the following expression (in the gas-in-gas case):

\[ F_{\text{diff}} = T_d D_{\text{gg}} S_g B_g (x_i - x_j) \]  

Eq. 3.55

where:

- \( T_d \) = diffusivity
- \( D_{\text{gg}} \) = gas-in-gas diffusion coefficient
- \( S_g \) = gas saturation
B_g = gas formation volume factor

x_i, x_j = the gas mole fraction in the gas phase in cells i and j.

More details can be found in “Diffusion” in the ECLIPSE Technical Description.

The diffusion coefficients should be entered in the appropriate unit set. The diffusion coefficients are often quoted in cm²/sec; they may be converted as follows:

1 cm²/s = 8.64 m²/day (METRIC)

1 cm²/s = 110.0976 ft²/day (FIELD)

1 cm²/s = 3600 cm²/hr (LAB)

Note: In 2014.1, the diffusion equation expressed in FIELD units has been modified. The pre 2014.1 behavior is restored by multiplying diffusion coefficients by 5.614583, which is the conversion factor between bbl and ft³. METRIC and LAB unit systems remain unchanged.

Examples

Example 1

With NTPVT=2:

```
DIFFC 100.0 1.217 1.2E-6 1.0E-7 5.0E-7 1.0E-8 /
      100.0 1.217 1.234E-6                       /
```

Example 2

With NTPVT=1.

Activate the cross phase diffusion option:

```
DIFFC 100.0 1.217 1.2E-6 1.0E-7 5.0E-7 1.0E-8 5.0E-7 1.0E-8 /
```
**Coal Bed Methane diffusion coefficients**

This keyword is used to specify the gas diffusion coefficient for each component in a coal bed methane run. A coal bed methane model can be divided into one or more regions. The number of regions is NTCREG and this number is input using REGDIMS item 6. Coal regions are input with the keyword COALNUM. There is one DIFFCBM record for each coal region up to NTCREG records. Each record contains the diffusion coefficients for the total number of components which is defined using keyword NCOMPS.

Record 1: \[d_1 \ d_2 \ d_3 \ldots \ d_{ncomps}\]

Diffusion coefficients for coal region number 1. The diffusion coefficients should be larger or equal to zero.

UNITs: m²/day (METRIC), ft²/day (FIELD), cm²/hr (LAB), m²/day (PVT-M)

DEFAULT: 0.0

Record 2: \[d_1 \ d_2 \ d_3 \ldots \ d_{ncomps}\]

Diffusion coefficients for coal region number 2.

Records 3 to NTCREG

The diffusion coefficients for coal regions 3 to NTCREG.

Each record is terminated by a slash (/). A final slash (/) terminates the keyword.

Diffusivities should be entered in the appropriate unit set. If the values available are in cm²/s, they may be converted as follows:

LAB units: 1 cm²/s = 3600 cm²/hr

METRIC or PVT-M units: 1 cm²/s = 8.64 m²/day

FIELD units: 1 cm²/s = 92.9979 ft²/day

For further information see "Coal bed methane model" in the ECLIPSE Technical Description.

**Example**

```
DIFFCBM
--For N2 CO2 CH4
    0.16 0.2 0.2 /
```

**Keywords**

605
Gas diffusion coefficients

This keyword specifies a gas phase diffusion coefficient for each component in a compositional run. These are used to define diffusive flows in terms of vapor mole fractions.

The normal diffusion coefficients input using by DIFFCGAS are defined by the condition (see Reid, Prausnitz and Polling [Ref. 90]):

\[ J_i = -cD_i \frac{\partial y_i}{\partial d} \]  

Eq. 3.56

where

- \( c \) is the total molar concentration.
- \( J_i \) is the flux of component \( i \) per unit area.
- \( D_i \) is the diffusion coefficient of component \( i \).
- \( \frac{\partial y_i}{\partial d} \) is molar concentration gradient of component \( i \).

These diffusion coefficients are used in ECLIPSE 300 to obtain gas interblock diffusive flows, which take the form:

\[ F_{\text{diff}} = T_D D_{\text{diff}} (S b^m_{\text{s}}) dy_i \]  

Eq. 3.57

where

- \( T_D \) is the diffusivity, the analogue of transmissibility for diffusive flow.
- \( y_i \) are vapor mole fractions.

For more details, see "Diffusion" in the ECLIPSE Technical Description.

Diffusivity should be entered in the appropriate unit set. If the values available are in \( \text{cm}^2/\text{s} \), they may be converted as follows:

- LAB units: 1 \( \text{cm}^2/\text{s} \) = 3600 \( \text{cm}^2/\text{hr} \)
- METRIC or PVT-M units: 1 \( \text{cm}^2/\text{s} \) = 8.64 \( \text{m}^2/\text{day} \)
- FIELD units: 1 \( \text{cm}^2/\text{s} \) = 92.9979 \( \text{ft}^2/\text{day} \)
- UNITS: \( \text{m}^2/\text{day} \) (METRIC), \( \text{ft}^2/\text{day} \) (FIELD), \( \text{cm}^2/\text{hr} \) (LAB), \( \text{m}^2/\text{day} \) (PVT-M)

DEFAULT: 0.0

Example

<table>
<thead>
<tr>
<th>DIFFCGAS</th>
<th>--For</th>
<th>N2</th>
<th>C1</th>
<th>C25</th>
<th>C7+1</th>
<th>C7+2</th>
<th>(field units)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>0.16</td>
<td>0.15</td>
<td>0.04</td>
<td>0.013</td>
<td>0.012</td>
<td>/</td>
</tr>
</tbody>
</table>
Gas-oil diffusion coefficients

This keyword specifies gas-oil cross phase diffusion coefficient for each component in a compositional run. These are used to define cross phase molecular diffusion from a cell containing gas to a cell containing oil. The opposite diffusive flow from a cell containing oil to a cell containing gas should be specified by the keyword DIFFCOG.

The potential is constructed by mixing the gas composition and the oil composition. The potential is then determined by using the liquid and or vapor mole fraction of the mixed cell.

For more details, see "Diffusion" in the ECLIPSE Technical Description.

Diffusivity should be entered in the appropriate unit set. If the values available are in cm²/s, they may be converted as follows:

<table>
<thead>
<tr>
<th>Units</th>
<th>Conversion Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>LAB</td>
<td>1 cm²/s = 3600 cm²/hr</td>
</tr>
<tr>
<td>METRIC or PVT-M</td>
<td>1 cm²/s = 8.64 m²/day</td>
</tr>
<tr>
<td>FIELD</td>
<td>1 cm²/s = 92.9979 ft²/day</td>
</tr>
<tr>
<td>UNITS</td>
<td>m²/day (METRIC), ft²/day (FIELD), cm²/hr (LAB), m²/day (PVT-M)</td>
</tr>
</tbody>
</table>

DEFAULT: 0.0

Example

```
DIFFCGO --For N2  C1  C25  C7+1  C7+2 (field units)
0.016  0.015  0.004  0.0013  0.0012 /
```
Gas-water diffusion coefficients

This keyword specifies gas-oil cross phase diffusion coefficient for each component in a compositional 
CO2STORE or GASWAT run. These are used to define cross phase molecular diffusion from a cell 
containing gas to a cell containing water. The opposite diffusive flow from a cell containing water to a cell 
containing gas should be specified by the keyword DIFFWG.

The potential is constructed by mixing the gas composition and the water composition. The potential is then 
determined by using the liquid and or vapor mole fraction of the mixed cell.

For more details, see "Diffusion" in the ECLIPSE Technical Description.

Diffusivity should be entered in the appropriate unit set. If the values available are in cm²/s, they may be 
converted as follows:

LAB units: 1 cm²/s = 3600 cm²/hr
METRIC or PVT-M units: 1 cm²/s = 8.64 m²/day
FIELD units: 1 cm²/s = 92.9979 ft²/day

UNITS: m²/day (METRIC), ft²/day (FIELD), cm²/hr (LAB), m²/day (PVT-M)

DEFAULT: 0.0

Example

<table>
<thead>
<tr>
<th>DIFFCGW</th>
<th>--For CO2 H2O (field units)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.016</td>
<td>0.015</td>
</tr>
</tbody>
</table>
**DIFFCOAL**

### Gas diffusion data

The data comprises NTCREG (see keyword **REGDIMS**, item 6) records, each terminated by a slash (/).

Each record consists of the following items of data:

1. The gas diffusion coefficient
   
   **UNITS:** \( \text{m}^2/\text{day} \) (METRIC), \( \text{ft}^2/\text{day} \) (FIELD), \( \text{cm}^2/\text{hr} \) (LAB)

2. The re-adsorption fraction
   
   **DEFAULT:** 1.0

3. The solvent diffusion coefficient
   
   **UNITS:** \( \text{m}^2/\text{day} \) (METRIC), \( \text{ft}^2/\text{day} \) (FIELD), \( \text{cm}^2/\text{hr} \) (LAB)
   
   **DEFAULT:** The gas diffusion coefficient (data item 1).

Setting the re-adsorption fraction to zero prevents re-adsorption.

The coal region number **COALNUM** is used to look up the diffusion data.

If **OPTIONS** keyword item 180 is activated, NTPVT (see keyword **TABDIMS**) records should be entered.

The region keyword **PVTNUM** is then used to look up the diffusion data.

### Example

With NTCREG = 2

```
DIFFCOAL
  0.2  
  0.1 0.0 / no re-adsorption
```

---

*x* ECLIPSE 100
  ECLIPSE 300
  SPECIAL
  RUNSPEC
  GRID
  EDIT
  PROPS
  REGIONS
  SOLUTION
  SUMMARY
  SCHEDULE

**Keywords**

609
Oil-gas diffusion coefficients

This keyword specifies oil-gas cross phase diffusion coefficient for each component in a compositional run. These are used to define cross phase molecular diffusion from a cell containing oil to a cell containing gas. The opposite diffusive flow from a cell containing gas to a cell containing oil should be specified by the keyword DIFFCGO.

The potential is constructed by mixing the oil composition and the gas composition. The potential is then determined by using the liquid and or vapor mole fraction of the mixed cell.

For more details, see "Diffusion" in the ECLIPSE Technical Description.

Diffusivity should be entered in the appropriate unit set. If the values available are in cm²/s, they may be converted as follows:

LAB units: 1 cm²/s = 3600 cm²/hr
METRIC or PVT-M units: 1 cm²/s = 8.64 m²/day
FIELD units: 1 cm²/s = 92.9979 ft²/day
UNITs: m²/day (METRIC), ft²/day (FIELD), cm²/hr (LAB), m²/day (PVT-M)
DEFAULT: 0.0

Example

```
DIFFCOG
--For      N2     C1      C25     C7+1    C7+2  (field units)
         0.16   0.15    0.04    0.013   0.012 /
```
Oil diffusion coefficients

This keyword specifies an oil phase diffusion coefficient for each component in a compositional run. These are used to define diffusive flows in terms of liquid mole fractions. Oil diffusion coefficients are typically an order of magnitude lower than gas diffusion coefficients.

The units and format of the DIFFCOIL keyword are entirely analogous to the DIFFGAS keyword.

UNITS: m²/day (METRIC), ft²/day (FIELD), cm²/hr (LAB), m²/day (PVT-M)

DEFAULT: 0.0
**DIFFCWAT  Water diffusion coefficients**

This keyword is specific to the `GASWAT`, `CO2STORE` and `GASSOL` options.

The keyword specifies water phase diffusion coefficients for each component in a compositional run. These are used to define diffusive flows in terms of liquid mole fractions. Water diffusion coefficients are typically an order of magnitude lower than gas diffusion coefficients.

The units and format of the `DIFFCWAT` keyword are entirely analogous to the `DIFFCGAS` keyword. Note that when `DIFFCWAT` is used in conjunction with the `GASSOL` option, one coefficient is specified for each component (including water) that is allowed to dissolve in the aqua phase.

**UNITS:** m$^2$/day (METRIC), ft$^2$/day (FIELD), cm$^2$/hr (LAB), m$^2$/day (PVT-M)

**DEFAULT:** 0.0
Water-gas diffusion coefficients

This keyword specifies water-gas cross phase diffusion coefficient for each component in a compositional CO2STORE or GASWAT run. These are used to define cross phase molecular diffusion from a cell containing water to a cell containing gas. The opposite diffusive flow from a cell containing gas to a cell containing water should be specified by the keyword DIFFCGW.

The potential is constructed by mixing the water composition and the gas composition. The potential is then determined by using the liquid and or vapor mole fraction of the mixed cell.

For more details, see “Diffusion” in the ECLIPSE Technical Description.

Diffusivity should be entered in the appropriate unit set. If the values available are in cm$^2$/s, they may be converted as follows:

LAB units: $1 \text{ cm}^2/\text{s} = 3600 \text{ cm}^2/\text{hr}$

METRIC or PVT-M units: $1 \text{ cm}^2/\text{s} = 8.64 \text{ m}^2/\text{day}$

FIELD units: $1 \text{ cm}^2/\text{s} = 92.9979 \text{ ft}^2/\text{day}$

UNITS: m$^2$/day (METRIC), ft$^2$/day (FIELD), cm$^2$/hr (LAB), m$^2$/day (PVT-M)

DEFAULT: 0.0

Example

```
DIFFCWG
--For CO2 H2O (field units)
0.16 0.15 /
```
DIFFDP

Restricts molecular diffusion in dual porosity runs

This keyword is relevant only in dual porosity runs (keyword DUALPORO in RUNSPEC) that use the molecular diffusion option (keyword DIFFUSE in RUNSPEC).

If this keyword is present, the molecular diffusion calculation is only performed for the matrix-fracture flows.

This can be useful to minimize the computer cost in dual porosity cases where the diffusive flows have been shown to be negligible except in the matrix-fracture flow, where diffusion may be the dominant recovery mechanism.

The keyword has no associated data.
Matrix-fracture diffusivity multipliers

This keyword is relevant only in dual porosity runs (keyword DUALPORO in the RUNSPEC section) which use either the molecular diffusion option (keyword DIFFUSE in the RUNSPEC section) or the coal bed methane option (keyword COAL in the RUNSPEC section).

The keyword should be followed by one non-negative real number for every grid block in the current input box. The values specified act as multipliers on the diffusivities calculated by ECLIPSE for the non-neighbor connections (NNCs) that represents the matrix-to-fracture flows in dual porosity runs. The DIFFMMF values from the upper half of the model (matrix cells) are used to modify the diffusivities; any values entered in the lower half (fracture cells) will be ignored. The data must be terminated by a slash (/).

Any DIFFMMF values that are not specified when the end of the GRID section is reached default to 1.0. If the keyword is specified in the SCHEDULE section then its effect will be cumulative on all values already specified or defaulted.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

Example

```
------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX       1 10 1 10 1 2 /
DIFFMMF   200*0.80 /
```
**DIFFMR**

**Diffusivity multipliers in the radial direction**

The keyword is used in radial geometry and is analogous to **DIFFMX** in Cartesian geometry.

It may only be used when the Molecular Diffusion option is activated (using the **DIFFUSE** keyword in the RUNSPEC section).

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the diffusivities calculated by the program for the +R faces of each grid block.

Thus, a value of **DIFFMR** specified for block (I, J, K) multiplies the diffusivity between blocks (I, J, K) and (I+1, J, K).

Any **DIFFMR** values that are not specified when the end of the GRID section is reached, default to 1.0.

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

Diffusivity multipliers are analogous to transmissibility multipliers for diffusive flow, and are described in "Diffusion" in the ECLIPSE Technical Description.

See also keyword **DIFFMR-**.

**Example**

```
---------- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX        3   5   1   10   2   4   /
DIFFMR     90*2.0 /
ENDBOX
```
**DIFFMR-**

**Diffusivity multipliers in the negative radial direction**

The keyword is used in radial geometry and is analogous to **DIFFMX-** in Cartesian geometry.

It may only be used when the Molecular Diffusion option is activated (using the **DIFFUSE** keyword in the **RUNSPEC** section).

In order to use **DIFFMR-**, the negative direction transmissibility multipliers must be enabled using the keyword **GRIDOPTS** in the **RUNSPEC** section.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the diffusivities calculated by the program for the -R faces of each grid block.

Thus, a value of **DIFFMR-** specified for block (I, J, K) multiplies the diffusivity between blocks (I, J, K) and (I-1, J, K).

Any **DIFFMR-** values that are not specified when the end of the **GRID** section is reached, default to 1.0.

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

See also the **DIFFMR** keyword.

If both **DIFFMR** and **DIFFMR-** are specified, then the product is used. For example, if **DIFFMR** is specified for cell (I, J, K) and **DIFFMR-** is specified for cell (I+1, J, K), then the product of the nominated **DIFFMR** and **DIFFMR-** values is used to multiply the diffusivity between blocks (I, J, K) and (I+1, J, K).

**Example**

```
  BOX  3   5   1  10   2   4   /
  DIFFMR-  90*2.0 /
  ENDBOX
```

---

**Keywords**

617
Diffusivity multipliers in the theta direction

The keyword is used in radial geometry and is analogous to DIFFMY in Cartesian geometry.

It may only be used when the Molecular Diffusion option is activated (using the DIFFUSE keyword in the RUNSPEC section).

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the diffusivities calculated by the program for the +THETA faces of each grid block.

Thus, a value of DIFFMTHT specified for block (I, J, K) multiplies the diffusivity between blocks (I, J, K) and (I, J+1, K).

Any DIFFMTHT values that are not specified when the end of the GRID section is reached, default to 1.0.

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

Diffusivity multipliers are analogous to transmissibility multipliers for diffusive flow, and are described in "Diffusion" in the ECLIPSE Technical Description.

See also keyword DIFFMTH-.

Example

```
-------- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX       3  5  1  10  2  4  /
DIFFMTHT   60*2.0 30*3.0 /
ENDBOX
```
**DIFFMTH-**

**Diffusivity multipliers in the negative theta direction**

The keyword is used in radial geometry and is analogous to DIFFMY- in Cartesian geometry.

It may only be used when the Molecular Diffusion option is activated (using the DIFFUSE keyword in the RUNSPEC section).

In order to use DIFFMTH-, the negative direction transmissibility multipliers must be enabled using the keyword GRIDOPTS in the RUNSPEC section.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the diffusivities calculated by the program for the -THETA faces of each grid block.

Thus, a value of DIFFMTH- specified for block (I, J, K) multiplies the diffusivity between blocks (I, J, K) and (I, J-1, K).

Any DIFFMTH- values that are not specified when the end of the GRID section is reached, default to 1.0.

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

See also the DIFFMTHT keyword.

If both DIFFMTHT and DIFFMTH- are specified, then their product is used. For example, if DIFFMTHT is specified for cell (I, J, K) and DIFFMTH- is specified for cell (I, J+1, K) then the product of the nominated DIFFMTHT and DIFFMTH- values is used to multiply the diffusivity between blocks (I, J, K) and (I, J +1, K).

**Example**

```
-------- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX       3  5  1 10  2  4  /
DIFFMTH-   60*2.0 30*3.0 /
ENDBOX
```

**Keywords**

619
**Diffusivity multipliers in the X-direction**

This keyword can only be used when the Molecular Diffusion option is activated (using the `DIFFUSE` keyword in the `RUNSPEC` section) in a Cartesian grid.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the diffusivities calculated by the program for the +X face of each grid block.

Thus, a value of `DIFFMX` specified for block (I, J, K) multiplies the diffusivity between blocks (I, J, K) and (I+1, J, K).

Any `DIFFMX` values that are not specified when the end of the `GRID` section is reached, default to 1.0.

If `DIFFMX` values are entered in the `GRID` section, they apply only to the diffusivities calculated by the program from information supplied in the `GRID` section, and **not** to any diffusivities entered directly in the `EDIT` section using the `DIFFX` keyword.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

Diffusivity multipliers are analogous to transmissibility multipliers for diffusive flow, and are described in "Diffusion" in the *ECLIPSE Technical Description*.

See also keyword `DIFFMX-`.

---

**Note:** Any non-neighbor connections generated due to faults will have diffusivities which reflect the `DIFFMX` values.

---

**Example**

```plaintext
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
   3   5   1  10   2   4   /
DIFFMX
90*2.0 /
ENDBOX
```
**DIFFMX-**

**Diffusivity multipliers in the negative X-direction**

This keyword can only be used when the Molecular Diffusion option is activated (using the `DIFFUSE` keyword in the `RUNSPEC` section) in a Cartesian grid.

In order to use `DIFFMX-`, the negative direction transmissibility multipliers must be enabled using the keyword `GRIDOPTS` in the `RUNSPEC` section.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the diffusivities calculated by the program for the -X face of each grid block.

Thus, a value of `DIFFMX-` specified for block (I, J, K) multiplies the diffusivity between blocks (I, J, K) and (I-1, J, K).

Any `DIFFMX-` values that are not specified when the end of the `GRID` section is reached, default to 1.0.

If `DIFFMX-` values are entered in the `GRID` section, they apply only to the diffusivities calculated by the program from information supplied in the `GRID` section, and **not** to any diffusivities entered directly in the `EDIT` section using the `DIFFX` keyword.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

See also the `DIFFMX` keyword.

If both `DIFFMX` and `DIFFMX-` are specified, then the product is used. For example, if `DIFFMX` is specified for the block (I, J, K) and `DIFFMX-` is specified for the block (I+1, J, K), then the program uses the product of the nominated `DIFFMX` and `DIFFMX-` values to multiply the calculated diffusivity between the adjacent blocks.

**Note:** Any non-neighbor connections generated due to faults will have diffusivities which reflect the `DIFFMX` and `DIFFMX-` values. For instance, a fault non-neighbor connection between a cell (I, J, K1) and a cell (I+1, J, K2) in an adjacent column will have a diffusivity multiplier of `DIFFMX` for block (I, J, K1) or `DIFFMX-` for block (I+1, J, K2), or the product of the two.

**Example**

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX      3      5     10      2     4     /
  DIFFMX- 90*2.0  /
  ENDBOX
```

**Keywords**

621
**DIFFMY**

**Diffusivity multipliers in the Y-direction**

This keyword can only be used when the Molecular Diffusion option is activated (using the `DIFFUSE` keyword in the `RUNSPEC` section) in a Cartesian grid.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the diffusivities calculated by the program for the +Y face of each grid block.

Thus, a value of `DIFFMY` specified for block (I, J, K) multiplies the diffusivity between blocks (I, J, K) and (I, J+1, K).

Any `DIFFMY` values that are not specified when the end of the `GRID` section is reached, default to 1.0.

If `DIFFMY` values are entered in the `GRID` section, they apply only to the diffusivities calculated by the program from information supplied in the `GRID` section, and **not** to any diffusivities entered directly in the `EDIT` section using the `DIFFY` keyword.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

Diffusivity multipliers are analogous to transmissibility multipliers for diffusive flow, and are described in "Diffusion" in the **ECLIPSE Technical Description**.

See also keyword `DIFFMY-`.

---

**Note:** Any non-neighbor connections generated due to faults will have diffusivities which reflect the `DIFFMY` values.

---

**Example**

```plaintext
------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  3 5 1 10 2 4 /
DIFFMY
  90*2.0 /
ENDBOX
```
**DIFFMY-**

**Diffusivity multipliers in the negative Y-direction**

This keyword can only be used when the Molecular Diffusion option is activated (using the `DIFFUSE` keyword in the RUNSPEC section) in a Cartesian grid.

In order to use `DIFFMY-`, the negative direction transmissibility multipliers must be enabled using the keyword `GRIDOPTS` in the RUNSPEC section.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the diffusivities calculated by the program for the -Y face of each grid block.

Thus, a value of `DIFFMY-` specified for block (I, J, K) multiplies the diffusivity between blocks (I, J, K) and (I, J-1, K).

Any `DIFFMY-` values which are not specified when the end of the GRID section is reached, default to 1.0.

If `DIFFMY-` values are entered in the GRID section, they apply only to the diffusivities calculated by the program from information supplied in the GRID section, and **not** to any diffusivities entered directly in the EDIT section using the `DIFFY` keyword.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

See also the `DIFFMY` keyword.

If both `DIFFMY` and `DIFFMY-` are specified, then the product is used. For example, if `DIFFMY` is specified for the block (I, J, K) and `DIFFMY-` is specified for the block (I, J+1, K), then the program uses the product of the nominated `DIFFMY` and `DIFFMY-` values to multiply the calculated diffusivity between the adjacent blocks.

**Note:** Any non-neighbor connections generated due to faults will have diffusivities which reflect the `DIFFMY` and `DIFFMY-` values. For instance, a fault non-neighbor connection between a cell (I, J, K1) and a cell (I, J+1, K2) in an adjacent column will have a diffusivity multiplier of `DIFFMY` for block (I, J, K1) or `DIFFMY-` for block (I, J+1, K2), or the product of the two.

---

**Example**

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX      3   5 10  2  4  /
DIFFMY-  90*2.0  /
ENDBOX
```
**DIFFMZ**

**Diffusivity multipliers in the Z-direction**

This keyword can only be used when the Molecular Diffusion option is activated (using the `DIFFUSE` keyword in the `RUNSPEC` section) in a Cartesian grid.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the diffusivities calculated by the program for the +Z face of each grid block.

Thus, a value of `DIFFMZ` specified for block (I, J, K) multiplies the diffusivity between blocks (I, J, K) and (I, J, K+1).

Any `DIFFMZ` values which are not specified when the end of the `GRID` section is reached, default to 1.0.

If `DIFFMZ` values are entered in the `GRID` section, they apply only to the diffusivities calculated by the program from information supplied in the `GRID` section, and **not** to any diffusivities entered directly in the `EDIT` section using the `DIFFZ` keyword.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

Diffusivity multipliers are analogous to transmissibility multipliers for diffusive flow, and are described in "Diffusion" in the *ECLIPSE Technical Description*.

See also keyword `DIFFMZ-`.

**Note:** Any non-neighbor connections generated due to pinchouts will have diffusivities which reflect the `DIFFMZ` values. Their behavior is analogous to the treatment of the transmissibility multipliers `MULTZ`.

**Example**

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
   3  5  1 10  2  4  /
DIFFMZ
90*2.0  /
ENDBOX
```
**DIFFMZ-**

**Diffusivity multipliers in the negative Z-direction**

This keyword can only be used when the Molecular Diffusion option is activated (using the `DIFFUSE` keyword in the RUNSPEC section) in a Cartesian grid.

In order to use `DIFFMZ-`, the negative direction transmissibility multipliers must be enabled using the keyword `GRIDOPTS` in the RUNSPEC section.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the diffusivities calculated by the program for the -Z face of each grid block.

Thus, a value of `DIFFMZ-` specified for block (I, J, K) multiplies the diffusivity between blocks (I, J, K) and (I, J, K-1).

Any `DIFFMZ-` values which are not specified when the end of the GRID section is reached, default to 1.0.

If `DIFFMZ-` values are entered in the GRID section, they apply only to the diffusivities calculated by the program from information supplied in the GRID section, and **not** to any diffusivities entered directly in the EDIT section using the `DIFFZ` keyword.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

See also the `DIFFMZ` keyword.

If both `DIFFMZ` and `DIFFMZ-` are specified, then the product is used. For example, if `DIFFMZ` is specified for the block (I, J, K) and `DIFFMZ-` is specified for the block (I, J, K+1), then the program uses the product of the nominated `DIFFMZ` and `DIFFMZ-` values to multiply the calculated diffusivity between the adjacent blocks.

**Note:** Any non-neighbor connections generated due to pinchouts will have diffusivities which reflect the `DIFFMZ-` values. Their behavior is analogous to the treatment of the transmissibility multipliers `MULTZ-`.

**Example**

```plaintext
---------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX

DIFFMZ- 3 5 1 10 2 4 /

/90*2.0 /

ENDBOX
```

Keywords
Diffusivities in the radial direction

The keyword specifies the diffusivity values in the radial directions, for runs using radial geometry. The values specified overwrite the diffusivities calculated by the program for the +R face of each grid block, when the molecular diffusion option is activated (using the \texttt{DIFFUSE} keyword in the \texttt{RUNSPEC} section) in a radial grid.

Thus, a value specified for block (I, J, K) is the diffusivity between blocks (I, J, K) and (I+1, J, K).

The \texttt{DIFFR} keyword is entirely optional. Any diffusivities that are not altered remain at the values calculated by the program from the \texttt{GRID} data.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

\textbf{UNITS:} m (\texttt{METRIC}), ft (\texttt{FIELD}), cm (\texttt{LAB}), m (\texttt{PVT-M})

\textbf{Note:} \texttt{DIFFR} does not affect non-neighbor connections due to faults.

\textbf{Note:} For ECLIPSE 100 compatibility, the \texttt{EDIT} section keyword \texttt{DIFFTR} has been aliased to \texttt{DIFFR}. However, \texttt{DIFFTY} can still be used in the \texttt{EDIT} section as documented in previous releases.

Diffusivities are analogous to transmissibilities for diffusive flow, and are described in "Diffusion" in the \textit{ECLIPSE Technical Description}.

\textbf{Example}

\begin{verbatim}
----------- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX
  1 1 1 3 1
DIFFR
  3*0.0
/
\end{verbatim}
DIFFTGAS  

**Thermal diffusion coefficients for gas**

This keyword specifies a gas phase thermal diffusion coefficient for each component in a compositional run. These are used to define gas phase diffusive flows in terms of temperature differences.

The gas phase diffusive flow is:

\[
F_{\text{diff}}^{ig} = T_D D_{ig}^a \left( \frac{y_i}{S_g} \frac{b_m}{RT} \right) A \left( RT \ln f_i - M_i G + M_i D_{ig}^T \ln T \right)
\]

Eq. 3.58

where

- \( T_D \) is the diffusivity, the analogue of transmissibility for diffusive flow,
- \( D_{ig}^a \) is the activity corrected gas diffusion coefficient, defined by the DIFFAGAS keyword,
- \( D_{ig}^T \) is the thermal gas diffusion coefficient, defined by this keyword, and
- \( T \) is the temperature.

For more details, see "Diffusion" in the ECLIPSE Technical Description.

There should be one value for each component.

UNITS: kJ/kg (METRIC), Btu/lb (FIELD), J/gm (LAB), kJ/kg (PVT-M)

DEFAULT: 0.0

**Example**

For 5 components:

```
DIFFTGAS
1.0 1.0 1.0 1.0 1.0/
```

**Note:** The keyword can only be used when the keyword DIFFAGAS is used, and in non-isothermal runs when the reservoir temperature is set with keyword TEMPVD or TEMPI.
**DIFFTHT**

**Diffusivities in the theta direction**

The keyword specifies the diffusivity values in the theta direction, for runs using radial geometry.

The values specified overwrite the diffusivities calculated by the program for the +Theta face of each grid block, when the molecular diffusion option is activated (using the `DIFFUSE` keyword in the `RUNSPEC` section) in a radial grid.

Thus, a value specified for block (I, J, K) is the diffusivity between blocks (I, J, K) and (I, J+1, K).

The `DIFFTHT` keyword is entirely optional. Any diffusivities which are not altered remain at the values calculated by the program from the `GRID` data.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

**UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

---

**Note:** `DIFFTHT` does not affect non-neighbor connections due to faults.

---

**ECLIPSE 300**

**Note:** For ECLIPSE 100 compatibility, the EDIT section keyword `DIFFTHT` has been aliased to `DIFFTHT`. However, `DIFFTY` can still be used in the EDIT section.

Diffusivities are analogous to transmissibilities for diffusive flow, and are described in “Diffusion” in the ECLIPSE Technical Description.

**Example**

```plaintext
------- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX       1 3 1 3 1 1 /
DIFFTHT   9*0.0 /
```

**Keywords**

628
**DIFFTOIL**

**Thermal diffusion coefficients for oil**

This keyword specifies an oil phase thermal diffusion coefficient for each component in a compositional run. These coefficients are used to define oil phase diffusive flows in terms of temperature differences. Oil diffusion coefficients are typically an order of magnitude lower than gas diffusion coefficients.

The units and format of the DIFFTOIL keyword are entirely analogous to the DIFFTGAS keyword. The keyword should only be used when the keyword DIFFAOIL is used, and in non-isothermal runs when the reservoir temperature is set with keyword TEMPVD or TEMPI.

There should be one value for each component.

**UNITS**: kJ/kg (METRIC), Btu/lb (FIELD), J/gm (LAB), kJ/kg (PVT-M)

**DEFAULT**: 0.0
Enables molecular diffusion

This indicates that the molecular diffusion option is required (see "Diffusion" in the ECLIPSE Technical Description).

In ECLIPSE 100, the diffusivities are calculated from the grid data. These calculated values can be modified using the DIFFMX and related keywords in the GRID section, or they can be overwritten using the DIFFX and related keywords in the EDIT section. The diffusion coefficients are entered using DIFFC in the PROPS section.

In ECLIPSE 300, the diffusivities are calculated from the grid data. These calculated values can be modified using the DIFFMX and related keywords in the GRID section, or they can be overwritten using the DIFFX and related keywords in the EDIT section. The diffusion coefficients are entered using DIFFCGAS or DIFFCOIL in the PROPS section.

The keyword has no associated data.
Diffusivities in the X-direction

The `DIFFX` keyword is entirely optional. This keyword specifies the diffusivity values explicitly, replacing those calculated by the program. Any diffusivities that are not altered remain at the values calculated by the program from the `GRID` data.

The values specified overwrite the diffusivities calculated by the program for the +X face of each grid block when the molecular diffusion option is activated (using the `DIFFUSE` keyword in the `RUNSPEC` section) in a Cartesian grid.

Thus, a value specified for block (I, J, K) is the diffusivity between blocks (I, J, K) and (I+1, J, K).

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted either side of the asterisk.

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

*Note:* `DIFFX` does not affect non-neighbor connections generated by faults.

---

Note: For ECLIPSE 100 compatibility, the EDIT section keyword `DIFFTX` has been aliased to `DIFFX`. However, `DIFFTX` can still be used in the EDIT section.

Diffusivities are analogous to transmissibilities for diffusive flow, and are described in "Diffusion" in the `ECLIPSE Technical Description`.

Example

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
   1 3 1 3 1 1 /
DIFFX
9*0.0 /
```

Keywords

631
Diffusivities in the Y-direction

The `DIFFY` keyword is entirely optional. This keyword specifies the diffusivity values explicitly, replacing those calculated by the program. Any diffusivities that are not altered, remain at the values calculated by the program from the `GRID` data.

The values specified overwrite the diffusivities calculated by the program for the +Y face of each grid block, when the molecular diffusion option is activated (using the `DIFFUSE` keyword in the `RUNSPEC` section) in a Cartesian grid.

Thus, a value specified for block (I, J, K) is the diffusivity between blocks (I, J, K) and (I, J+1, K).

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

**UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

---

**Note:** `DIFFY` does not affect non-neighbor connections generated by faults.

---

**Note:** For ECLIPSE 100 compatibility, the `EDIT` section keyword `DIFFTY` has been aliased to `DIFFY`. However, `DIFFTY` can still be used in the `EDIT` section.

Diffusivities are analogous to transmissibilities for diffusive flow, and are described in "Diffusion" in the *ECLIPSE Technical Description*.

**Example**

```
----------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
   1 3 1 3 1 1 /
DIFFY
   9*0.0 /
```
DIFFZ

Diffusivities in the Z-direction

The DIFFZ keyword is entirely optional. This keyword specifies the diffusivity values explicitly, replacing those calculated by the program. Any diffusivities that are not altered, remain at the values calculated by the program from the GRID data.

The values specified overwrite the diffusivities calculated by the program for the +Z face of each grid block, when the molecular diffusion option is activated (using the DIFFUSE keyword in the RUNSPEC section).

Thus, a value specified for block (I, J, K) is the diffusivity between blocks (I, J, K) and (I, J, K+1).

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVTV-M)

Diffusivities are analogous to transmissibilities for diffusive flow, and are described in "Diffusion" in the ECLIPSE Technical Description.

Example

```
BOX
  1 3 1 3 1 2 /
DIFFZ
  18*0.0 /
```
**DIMENS**

** Specifies the dimensions of the grid **

This keyword defines the basic size of the simulation grid. It is followed by three integers, specifying the number of cells in the x, y and z directions respectively. (In the radial case, these values specify the number of cells in the r, θ and z directions). In the documentation, the items in the DIMENS keyword are frequently referred to as $N_x$, $N_y$ and $N_z$, or NDIVIX, NDIVIY and NDIVIZ.

There is no default. It is an error not to supply the grid dimensions.

IN ECLIPSE 100, when using any discretized matrix option (keyword NMATRIX) including the discretized gravity drainage model activated by keywords GRAVDRB and NMATOPTS (option VERTICAL), the minimum value of the number of cells in the X direction is 2.

**Example**

For a reservoir represented by a 10 x 3 x 4 grid the keyword would be:

```
DIMENS
10 3 4 /
```
DIMPES

**Dynamically sets IMPES solution procedure**

The DIMPES keyword switches from the default implicit solution technique to the IMplicit Pressure Explicit Saturation solution procedure.

The data comprises the following data items terminated by a slash (/). If a slash is encountered before the end of the data, the remaining items are left at their default values.

**Data items**

- **DSTARG** Target maximum saturation change in any timestep  
  DEFAULT: 0.05

- **DSMAX** Maximum tolerable saturation change in any timestep  
  DEFAULT: 0.1

- **DPMAX** Maximum tolerable pressure change in any timestep  
  UNITS: bars (METRIC), psi (FIELD), atm (LAB)  
  DEFAULT: 200 psi

**Examples**

**Example 1**

To set the default values explicitly:

```
DIMPES /  
```

**Example 2**

To set the maximum target saturation change in any grid block in any timestep to 2.5% and the maximum tolerable saturation change to 5%. ECLIPSE attempts to select timesteps such that the maximum saturation change in any grid block is 2.5% but it accepts a timestep if the maximum saturation change is less than 5%.

```
DIMPES 0.025 0.05 /  
```
**DIMPLICT**

**Resets the solution procedure to fully implicit**

This keyword requires no data and can be used to switch from the IMPES method back to the Fully Implicit method (see keywords IMPES, DIMPES and FULLIMP). The TUNING parameters are reset to their default values.

This was formerly the **IMPLICIT** keyword in ECLIPSE 100.

*Note:* The DIMPLICT keyword cannot be used if the IMPES keyword has been specified in the RUNSPEC section. If the base case of a restart run was run with IMPES, however defined, the restart run cannot use the DIMPLICT keyword.

### Example

```
DIMPLICT
```

DISGAS

Run contains dissolved gas in live oil

This indicates that the run contains dissolved gas in live oil.

The keyword may be used only if OIL and GAS are both present.

If oil has a constant and uniform dissolved gas concentration, and if the pressure never drops below the bubble point, then the model can be run more efficiently by omitting the GAS and DISGAS keywords from the RUNSPEC section, treating the oil as dead oil, and declaring a constant Rs value with keyword RSCONST or RSCONSTT in the PROPS section. The simulation is run as a dead oil problem with no gas, but the calculations take into account the constant dissolved gas concentration.

The keyword has no associated data.

When the BLACKOIL keyword is used in ECLIPSE 300 this option is automatically selected if the PVTO or PVCO keywords are present.
**DISPBC**

**Velocity boundary conditions for coupled rock and stress calculations**

This keyword is being replaced by new keywords for setting displacement boundary conditions. Please refer to keyword **GMDISBC**.

This keyword specifies displacement boundary conditions when a coupled rock stress and fluid flow calculation is specified with the keyword **GEOMECH** in the **RUNSPEC** section.

This keyword allows displacement boundary conditions to be specified on all boundaries. If a particular boundary is not specified in this keyword or by **TRACTBC** or **PSTRBC** keywords, then that boundary is defaulted to a rigid boundary.

This keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the velocity boundary condition is applied.
   
   This must be one of the following:
   
   - I+ Specify the boundary condition on the outer I+ edge
   - I- Specify the boundary condition on the outer I- edge
   - J+ Specify the boundary condition on the outer J+ edge
   - J- Specify the boundary condition on the outer J- edge
   - K+ Specify the boundary condition on the outer K+ edge
   - K- Specify the boundary condition on the outer K- edge

   DEFAULT: There is no default for this item; it must be specified.

   The K- boundary is the top boundary of the simulation, K+ is the bottom.

2. An integer specifying the type of boundary condition on this face

   This must be one of the following:

   1. A normal rock displacement velocity is specified on the boundary denoted in item 1.

   2. A rock displacement velocity with X, Y, Z components. A single X, Y, Z velocity component is set in item 3 and the component name must be set in item 4. Separate lines are needed to set different components.

   For boundary condition type 2, the X, Y, Z components conform to the X, Y, Z directions as specified in the **COORD** keyword for corner point geometry.

   DEFAULT: 2
3. A normal rock displacement on this boundary, if the boundary condition type in item 2 is 1, or an X, Y, Z component of a rock displacement, if the boundary condition type in item 2 is 2.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
DEFAULT: 0

4. A character string that gives the component of displacement or velocity if the boundary type in item 2 is 2. This must be one of ‘X’, ‘Y’ or ‘Z’.
DEFAULT: None. It must be specified if item 2 is 2

5. First block to be modified in the I direction
DEFAULT: 1 if an I- face

6. Last block to be modified in the I direction
DEFAULT: NX if an I+ face

7. First block to be modified in the J direction
DEFAULT: 1 if a J- face

8. Last block to be modified in the J direction
DEFAULT: NY if a J+ face

9. First block to be modified in the K direction
DEFAULT: 1 if a K- face

10. Last block to be modified in the K direction
DEFAULT: NZ if a K+ face

Example

```
-- set a normally rigid boundary in the x and y-directions
DISPBC
-- side type velocity
‘I+’ 1 /
‘I-’ 1 /
‘J+’ 1 /
‘J-’ 1 /
```

Keywords
Velocity boundary conditions for coupled rock and stress calculations in a Local Grid Refinement

This keyword is being replaced by new keywords for setting displacement boundary conditions. Please refer to keyword GMDISBCL.

This keyword specifies displacement boundary conditions when a coupled rock stress and fluid flow calculation is specified with the keyword GEOMECH in the RUNSPEC section.

This keyword allows displacement boundary conditions to be specified on all boundaries of a local grid refinement. If a particular boundary is not specified by this keyword or by TRACTBC or PSTRBC keywords, then that boundary is defaulted to a rigid boundary.

This keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n* where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the velocity boundary condition is applied

   This must be one of the following:

   I+  Specify the boundary condition on the outer I+ edge

   I-  Specify the boundary condition on the outer I- edge

   J+  Specify the boundary condition on the outer J+ edge

   J-  Specify the boundary condition on the outer J- edge

   K+  Specify the boundary condition on the outer K+ edge

   K-  Specify the boundary condition on the outer K- edge

   DEFAULT: There is no default for this item; it must be specified.

   The K- boundary is the top boundary of the simulation, K+ is the bottom.

2. An integer specifying the type of boundary condition on this face

   This must be one of the following:

   1. A normal rock displacement velocity is specified on the boundary denoted in item 1.

   2. A rock displacement velocity with X, Y, Z components. A single X, Y, Z velocity component is set in item 3 and the component name must be set in item 4. Separate lines are needed to set different components.

      For boundary condition type 2, the X, Y, Z components conform to the X, Y, Z directions as specified in the COORD keyword for corner point geometry.
DEFAULT: 2

3. Name of the Local Grid Refinement

4. A normal rock displacement on this boundary, if the boundary condition type in item 2 is 1, or an X, Y, Z component of a rock displacement, if the boundary condition type in item 2 is 2.

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

   DEFAULT: 0

5. A character string that gives the component of displacement or velocity if the boundary type in item 2 is 2. This must be one of ‘X’, ‘Y’ or ‘Z’.

   DEFAULT: None. It must be specified if item 2 is 2

6. First block to be modified in the I direction in an LGR specified in item 3

   DEFAULT: 1 if an I- face

7. Last block to be modified in the I direction in an LGR specified in item 3

   DEFAULT: NX if an I+ face

8. First block to be modified in the J direction in an LGR specified in item 3

   DEFAULT: 1 if a J- face

9. Last block to be modified in the J direction in an LGR specified in item 3

   DEFAULT: NY if a J+ face

10. First block to be modified in the K direction in an LGR specified in item 3

    DEFAULT: 1 if a K- face

11. Last block to be modified in the K direction in an LGR specified in item 3

    DEFAULT: NZ if a K+ face

Example

```
-- set a normally rigid boundary in the x and y-
directions for the local grid refinement LGR1
DISPBCL
   -- side type velocity
   'I+' 1 'LGR1' /
   'I-' 1 'LGR1' /
   'J+' 1 'LGR1' /
   'J-' 1 'LGR1' /
   / */
```

Keywords

641
Dispersion dimension data

The data consists of the following items, describing the dimensions of the dispersion tables to be used in the run (see TRDIS). The data must be terminated by a slash (/).

The items are:

1. The number of dispersion tables
   DEFAULT: 1
2. The maximum number velocity nodes in any dispersion table (there must be a minimum of 2)
   DEFAULT: 2
3. The maximum number of concentration nodes per velocity node in any dispersion table.
   DEFAULT: 1

See also TRDIS, DISPERSE and DSPDEINT.

Example

```
DISPDIMS
9 4 4 /
```
Dispersion tables

The data comprises MXDIST (see item 1 in keyword DISPDIMS in the RUNSPEC section) tables of dispersion coefficients (see TRDIS). A single table consists of at least 2 and up to MXVVPT (item 2 in keyword DISPDIMS) records, each terminated by a slash (/). A table is terminated by a null record (that is a record with no data before the terminating slash). Most records contain the following items of data, terminating with a slash.

1. Velocity (absolute value)
   The values must be monotonically increasing.
   
   UNITS: m/day (METRIC), ft/day (FIELD), cm/hr (LAB)

2. Tracer concentration

3. Dispersion coefficient.
   
   UNITS: m^2/day (METRIC), ft^2/day (FIELD), cm^2/hr (LAB)

The first velocity in the table must be zero.

However, some records (optionally all) contain additional data that defines the dispersion coefficient, at a given velocity, as a function of tracer concentration. For each velocity the first tracer concentration value must always be zero. The additional data takes the form of 2 columns which continue from items 2 and 3 above. There can only be a maximum of MXCVPV (item 3 in keyword DISPDIMS) concentration nodes per velocity node. The concentration values must be monotonically increasing.

**Note:** For a brine tracer, item 2 can be replaced with water density if the DSPDEINT keyword is used.

**Note:** There is no need for the dispersion coefficient to be monotonically increasing or decreasing, and it is perfectly allowable to use constant values (two velocity nodes will still be needed). A particularly common table will have 2 velocity nodes, both with a zero dispersion coefficient to effectively turn off certain terms (see TRDIS). Non-zero constants can be used to enable heterogeneous diffusion. If dispersion is only being used to enable a non-homogeneous diffusion then the calculations can be significantly speeded up by using the TRNHD keyword.

Example

```
DISPERSE
  0  0.0  0.000523
  1.0  0.000521  /
  90  0.0  0.000132
  1.0  0.000131  /
 150  0.0  0.0000877
  1.0  0.0000861  /
 210  0.0  0.0000554
  1.0  0.0000555  /
 270  0.0  0.0000417
  1.0  0.0000421  /
 330  0.0  0.0000357
  1.0  0.0000361  /
 530  0.0  0.0000356
  1.0  0.0000360  /
/ null record to terminate table 1
  0  0.0  0.000523
  120  0.0  0.000132
  180  0.0  0.0000877
  240  0.0  0.0000554
```

Keywords

643
| 300 | 0.0 | 0.0000417 | / |
| 360 | 0.0 | 0.000000357 | / |
| 560 | 0.0 | 0.00356 | / No dependence on concentration |
| / null record to terminate table 2 |
Partial densities for NGLs

In a compositional run using gas plants which produce NGLs, that is natural gas liquids (specified using GPTABLEN), this keyword specifies the partial densities of the NGLs. These are converted to partial molar volumes internally, and the molar volume of a NGL mixture is obtained as a mole fraction weighted sum. Note that for a volatile component, an apparent partial density should be entered.

See also "Gas Plants and NGLs" in the ECLIPSE Technical Description.

The keyword should be followed by $N_c$ values.

*UNITS: kg/m$^3$ (METRIC), lb/ft$^3$ (FIELD), gm/cc (LAB), kg/m$^3$ (PVT-M)*

**Example**

```
DNGL
50.192  48.507  61.991  26.532  34.211  36.333
34.772  36.146  38.705  39.080
```
**DOMAIN**

**Set non-default domain size for phase property evaluation**

This keyword may be used to override the default treatment of domains used for phase property evaluation.

The active cells within a reservoir are split into domains of cells, which are treated together in the phase equilibrium and property evaluation parts of the code.

Reducing the number of cells in a domain slightly reduces the memory used, as the workspace required is reduced, at the expense of vector length.

It is inadvisable to use very small domain sizes (below 50) as the loop and subroutine calling overheads become significant.

The actual domain size used will, in any event, never be greater than the number of cells in the problem.

If the Local Grid Refinement option is active then this keyword can be used in conjunction with the CARFIN keyword to alter the partitioning of a CARFIN that is being processed in parallel. In this case the LGR must have been flagged to run in parallel and the DOMAIN keyword must appear between the CARFIN and the corresponding ENDFIN. The DOMAIN keyword should NOT be used if the refinement is part of an amalgamation.

**Example**

```
DOMAIN
   100 /
```
**DOMAINS**

**Modifies the size of parallel domains**

This keyword is optional, and allows the user to modify the number of planes allocated to each domain.

The keyword should be followed by one positive real number for each domain in the model. The number of domains is specified using keyword **PARALLEL** in the RUNSPEC section. The data must be terminated by a slash. The total number of planes must sum to the number of cells in the outer solver direction. The solver direction sequence determined by ECLIPSE can be output using the **DOMAINS** mnemonic in the **RPTGRID** keyword. However, it is advisable to set the outer solver direction manually using the **SOLVDIRS** keyword when using this keyword.

**Example**

Outer direction X, NX=40 and NDMAIN=8

```
SOLVDIRS
 '-X' /
DOMAINS
 7 6 4 3 3 6 7 /
```
DPCDT

Maximum rate of change of capillary pressure

The data consists of a single item:

1. The maximum rate of change of capillary pressure in explicit cells.
   
   **UNITS:** bars/day (METRIC), psi/day (FIELD), atm/hr (LAB), atm/day (PVT-M)
   
   **DEFAULT:** 0.0099972 psi/day in IMPES and AIM runs, no limit in fully implicit runs.
   
   Rather than freezing the capillary pressure we limit the rate at which it can change from step to step. This stabilizes the run without seriously affecting the modeling of imbibition driven recovery. In the implicit cells in AIM cases this limit can cause convergence problems, in these cells the limit is replaced by a limit of the rate of change with respect to saturation.
   
   By default IMPES and AIM modes have a maximum rate of change of capillary pressure $\frac{dP_c}{dt} = 0.0099972$ psi/day. This limit can be reset using the DPCDT keyword. The option of freezing the capillary pressures is retained through the FREEZEPC keyword. This has the same effect as setting DPCDT to 0 in IMPES mode.
   
   The DPCDT limit may be disabled by using the NODPCDT keyword.

**Note:** By default, if you are using the CFLLIMIT keyword, then no limit is applied to the capillary pressure change.

**Example**

Allow capillary pressure to change at up to 1 psi/day:

```
DPCDT 1 /
```
Use matrix cell grid data for fracture cells

This keyword may be used to simplify the input of data for dual porosity runs. It enables grid data to be entered for the matrix cells only (the first \(\text{NDIVIZ}/2\) layers), the missing values for the remaining fracture layers being obtained from the corresponding matrix cell.

This operation will be performed for the \(\text{DX}, \text{DY}, \text{DZ}, \text{PERMX}, \text{PERMY}, \text{PERMZ}, \text{PORO}, \text{TOPS}, \text{MIDS}, \text{NTG}, \text{DZNET}, \text{ZCORN}, \text{PERMXY}, \text{PERMYZ}\) and \(\text{PERMZX}\) keywords. The copying of the matrix cell value into the fracture cell value will only occur if the fracture cell value has not been defined by the user.

In the case of the \(\text{ZCORN}\) keyword, only half the normal set of corner point depths need be supplied (and generated by PETREL). This is useful when preparing faulted dual porosity runs. As in the case of the other keywords, the \(\text{BOX}\) keyword may be used to specify values for the cells required.

When using an unstructured grid supplied by FloGrid, the \(\text{DEPTHs}\) entered in the \(\text{EDIT}\) section are also copied from matrix to fracture.

When using the Temperature Option, the thermal conductivity \(\text{THCONR}\) is also copied from the matrix to the fracture.

When using the Thermal Option, the thermal conductivity \(\text{THCONR}\), and the rock heat capacity data \(\text{HEATCR}\) and \(\text{HEATCRT}\) are copied from the matrix to the fracture.

When using the Thermal Option, the rock thermal conductivity \(\text{THCROCK}\), and the phase thermal conductivities \(\text{THCOIL}, \text{THCGAS}, \text{THCWATER}\) or \(\text{THCSOLID}\) are also copied from the matrix to the fracture.

This keyword contains no items.
DPKRMOD  Modifies the matrix oil relative permeability in dual porosity cases

This keyword is optional in dual porosity runs, and gives you control of the matrix relative permeability. The keyword provides a simple way of modifying the oil relative permeability to help match a single porosity model of a single matrix block, and optionally allows the fracture to matrix flow to reflect the maximum relative permeability in the matrix.

The keyword should be followed by NTSFUN (see keyword TABDIMS in the RUNSPEC section) records, each consisting of three items of data terminated by a slash (/):

1. The modification parameter \( m_w \) for oil-in-water relative permeability.
   The value should be between -1.0 and 1.0.
   DEFAULT: 0.0

2. The modification parameter \( m_g \) for oil-in-gas relative permeability.
   The value should be between -1.0 and 1.0
   DEFAULT: 0.0

3. Scale the relative permeabilities for fracture-to-matrix flow, to reflect the maximum relative permeability at the residual saturation of the displaced phase?
   YES  Modify the fracture relative permeabilities for fracture-matrix flow.
   NO   No modification to the fracture relative permeabilities.
   DEFAULT: NO

The relative permeability curves are modified by applying a quadratic multiplier function, such that the end-points \( (S_{OCR}, 1-S_{WCO}) \) are preserved and the relative permeability at a saturation half way between the end-points is invariant.

The modification parameter is not a physical quantity and should be regarded only as a tuning parameter to match a dual porosity block with a finely gridded single porosity model.

If the modification parameter = 0.0, then no modification occurs.
If the modification parameter > 0.0, then the \( K_r \) at low oil saturation is boosted and the \( K_r \) at high oil saturation is reduced relative to the original curve.
If the modification parameter < 0.0, then the \( K_r \) at low oil saturation is reduced and the \( K_r \) at high oil saturation is increased relative to the original curve.

The scaling for oil-in-water relative permeability is as follows for the case where \( m_w \) is greater than 0.0:

\[
K_r = \tilde{K}_r + M \left( K_r(s) - \tilde{K}_r \right)
\]

Eq. 3.59

where

\( \tilde{K}_r \) is the relative permeability at \( \tilde{s} = \frac{SOWCR + 1.0 - SWCO}{2} \)

\( K_r(s) \) is the input table relative permeability
SOWCR is the critical oil to water saturation
SWCO is the connate water saturation

\[ M = 4.0 m_w K_r (s')^2 - 4.0 m_w K_r (s') + 1 \]

If \( m_w \) is less than 0.0, then the same scaling is applied but with the X and Y axes \((s, K_r (s))\) reversed:

For a given saturation \( s \), a new saturation \( s' \) is calculated such that:

\[ \tilde{s} + M (s' - \tilde{s}) = s \]  \hspace{1cm}  \text{Eq. 3.60}

where

\[ \tilde{s} = (\text{SOWCR} + 1.0 - \text{SWCO}) / 2 \]

\( M \) is the modification function, which is given by \( M = 4.0 m_w K_r (s')^2 - 4.0 m_w K_r (s') + 1 \)

\( K_r \) is then taken as \( K_r (s') \).

The third data item initiates scaling of the fracture relative permeability curves when calculating the fracture to matrix flow only. The fracture relative permeability is scaled such that the maximum relative permeability is that of the matrix at the residual saturation of the displaced phase. For example in the water case:

\[ K_f (S_w) = K_{table} (S_w) \cdot K_{rmmax} / K_{rfmax} \]  \hspace{1cm}  \text{Eq. 3.61}

where

\[ K_{table} (S_w) \] is the relative permeability looked up using the input table

\( K_{rmmax} \) is the matrix relative permeability at \( S_w = 1.0 - S_{owcr} \)

\( K_{rfmax} \) is the maximum fracture relative permeability

\( S_{owcr} \) is the critical oil saturation to water.

See "Dual porosity" in the ECLIPSE Technical Description.

Example

```
  DPKRMOD 0.0 0.9 /       Table 1 Matrix
           /             Table 2 Fracture (default)
  --- Request the modified fracture/matrix relative permeability
  DPKRMOD 2* YES /
  2* YES /
```

Keywords
The DPNUM keyword may be used for a dual porosity, single permeability run to specify regions within the reservoir to be treated as single porosity only.

The keyword should be followed by the integer 0 or 1 for each grid block in the first \( \text{NDIVIZ}/2 \) layers, to indicate whether the grid block should be modeled as single porosity (0) or dual porosity (1). Any grid blocks that do not have a DPNUM value entered default to dual porosity (1).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

Grid data for single porosity cells is only required in the first \( \text{NDIVIZ}/2 \) layers. That is the grid block properties are only required for the matrix cells; the fracture cells are made inactive and ignored.

Within single porosity regions, flow is between matrix blocks. Wells may only connect with the matrix blocks in single porosity regions.

The keyword is ignored unless dual porosity, single permeability is enabled in the RUNSPEC section using keyword DUALPORO. Single porosity regions specified by the keyword DPNUM are not allowed in a dual permeability run, so the keyword DUALPERM must not be specified in this case.

This keyword is not permitted if multi porosity is specified, either as a general multi porosity run using N MATRIX with value greater than 1, or as a triple porosity run using TRPLPORO.

For ECLIPSE 100, there are technical problems if all the DPNUM values in the grid are set to zero, so this is prohibited; it is not possible to use DPNUM in this way to switch between single and dual porosity.

For LGRs in ECLIPSE 100, all the DPNUM values in the local grid must be the same, so that the LGR is either single or dual porosity. The values are specified for the region of the global grid occupied by the LGR, not in the CARFIN or RADFIN block.

For ECLIPSE 300, DPNUM cannot be used when alternative transmissibility multipliers are used, that is, when the first item of the GRIDOPTS keyword is set to ‘YES’.

Example

To specify an 8 x 4 x 4 run as dual porosity in half of the reservoir, specify DUALPORO in RUNSPEC, and add the following in the GRID section:

```
   4*0  4*1  4*0  4*1
   4*0  4*1  4*0  4*1
   4*0  4*1  4*0  4*1
   4*0  4*1  4*0  4*1 /
```
The keyword should be followed by one positive real number for every grid block in the current input box specifying its size in the R direction. The data must be terminated by a slash (/).

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

Note: The inner radius of the innermost (I = 1) grid blocks should be set using keyword INRAD. The outer radius of these blocks is equal to the sum of the inner radius and DR.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

In ECLIPSE 100, every DR value in the top plane (K = 1) must be specified in one way or another by the end of the GRID section. Values in lower planes (K > 1) that are not specified default to the value in the plane above.

Example

```
--------- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX      1 12 1 6 1 1
INRAD    1
DR
 1  2  4  8 16 32  64 128  256 512 1000 2000
 1  2  4  8 16 32  64 128  256 512 1000 2000
 1  2  4  8 16 32  64 128  256 512 1000 2000
 1  2  4  8 16 32  64 128  256 512 1000 2000
 1  2  4  8 16 32  64 128  256 512 1000 2000
```
DRAINAGE  Request drainage option in hysteresis

This option restricts \( Kr \) values obtained in the hysteresis option to lie on or below drainage curve. This is not always obtained in the Killough relative permeability hysteresis option.

This is not a problem with the Jargon hysteresis option. **HYSTK** or **HYST** is used to enable the Killough relative permeability hysteresis option. **HYSTJ** is used to enable the Jargon option.

If the **ODD3P** relative permeability and capillary pressure hysteresis model has been selected this keyword will be ignored.

**Keywords**

<table>
<thead>
<tr>
<th>ECLIPSE 100</th>
<th>ECLIPSE 300</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>SPECIAL</td>
</tr>
<tr>
<td>RUNSPEC</td>
<td></td>
</tr>
<tr>
<td>GRID</td>
<td></td>
</tr>
<tr>
<td>EDIT</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>PROPS</td>
</tr>
<tr>
<td>REGIONs</td>
<td></td>
</tr>
<tr>
<td>SOLUTION</td>
<td></td>
</tr>
<tr>
<td>SUMMARY</td>
<td></td>
</tr>
<tr>
<td>SCHEDULE</td>
<td></td>
</tr>
</tbody>
</table>
Reference densities

In a run with \( N \) components this keyword specifies the fluid densities at the reference pressures and temperatures. The keyword can be used either in a run which uses the Zudkevitch Joffe equation, or in a Thermal run (keyword THERMAL in the RUNSPEC section).

In a live-oil thermal run, the oil component liquid densities are given by:

\[
\rho_c = \left[ 1 + c_{T1c} \left( T - T_{ref_c} \right) \right] \left[ 1 - c_{Pc} \left( \frac{P - P_{ref_c}}{P_{ref_c}} \right) \right] \rho_{ref_c}
\]

Eq. 3.62

where

- \( \rho_{ref_c} \) is the reference density, specified with this keyword
- \( c_{Pc} \) is the oil component liquid compressibility, specified with keyword CREF
- \( P_{ref_c} \) is the reference pressure, specified with keyword PREF
- \( T_{ref_c} \) is the reference temperature, specified with keyword TREF
- \( c_{T1c} \) is the thermal expansion coefficient, specified with keyword THERMEX1.

There should be one value for each component:

UNITs: kg/m\(^3\) (METRIC), lb/ft\(^3\) (FIELD), gm/cc (LAB), kg/m\(^3\) (PVT-M)

DEFAULT: 80\% of the density of water.

DREF is an alternative to GREF.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

**Example**

```
DREF
50.192 48.507 61.991 26.532 34.211 36.333
34.772 36.146 38.705 39.080 41.140 42.701
43.887 44.823 45.572 46.197 52.285 56.632 /
```
DREFS

Reference densities for surface EoS

This optional keyword supplies reference densities for use in the surface calculations. The default, if this is not supplied, is for these to follow the reservoir values supplied with DREF.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.
**DREFT**

**Reference densities**

This keyword can only be used with the Thermal option (keyword **THERMAL** in the **RUNSPEC** section).

**DREFT** is an alias for **DREF**. This keyword was introduced to facilitate export of thermal data from PVT $i$. It makes no difference to the simulator whether this keyword or the older **DREF** keyword is used.
**DREFTS**

**Reference densities for surface EoS**

This keyword can only be used with the Thermal option (keyword `THERMAL` in the `RUNSPEC` section).

**DREFTS** is an alias for `DREFS`. This keyword was introduced to facilitate export of thermal data from PVT $i$. It makes no difference to the simulator whether this keyword or the older `DREFS` keyword is used.
DREFW specifies the water density at a reference pressure for each of the $N_w$ water components.

The water component densities, at pressure $P$, are calculated from:

$$
\rho_w = \rho_{ref w} \cdot (1 + X + X^2 / 2)
$$

$$
X = c_P \cdot (P - P_{ref w})
$$

where

$\rho_{ref w}$ is the reference density, specified with this keyword

$c_P$ is the water component compressibility, specified with keyword CREFW

$P_{ref w}$ is the reference pressure, specified with keyword PREFW

There should be one value for each component:

UNITs: kg/m$^3$ (METRIC), lb/ft$^3$ (FIELD), gm/cc (LAB), kg/m$^3$ (PVT-M)

DEFAULT: 999.014 kg/m$^3$

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

See "Water components" in the ECLIPSE Technical Description for further details of the water component option.

**Example**

```
DREFW
1000.0 1010.0 /
```
DREFWS  Reference densities for water components for surface EoS

This optional keyword supplies reference densities of water components for use in the surface calculations. The default, if this is not supplied, is for these to follow the reservoir values supplied with DREFW.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation-of-state regions is entered as item 10 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.
Defines the default priority formula for the prioritized drilling queue

This keyword is used to set the coefficients that define the default priority formula for the prioritized drilling queue. It must be used if any wells are placed in this queue with the WDRILPRI keyword without a fixed priority set in WDRILPRI item 2.

Wells are opened from the drilling queue whenever they are needed to maintain a group rate target under group control by guide rate (GCONPROD, GCONINJE, GCONSAL). They may also be opened from the drilling queue in ECLIPSE 100 should they be needed to maintain a group’s production potential (GDRILPOT). The drilling queue may either be a sequential drilling queue constructed with keyword QDRILL, or a prioritized drilling queue constructed with keywords WDRILPRI and DRILPRI. In a sequential drilling queue wells are opened in the sequence in which they were placed in the queue. In a prioritized drilling queue wells are opened in decreasing order of their drilling priority. In ECLIPSE 100 the order of opening may, however, be affected by the availability of drilling rigs (see keywords WDRILTIM and GRUPRIG). If multiple wells have the same highest drilling priority, the well which is first defined by WELSPECS or WELSPECL will be drilled first. Only one type of drilling queue may exist at any given time in the run; the two types of queue cannot operate together.

Drilling priorities for wells in the prioritized drilling queue are calculated at regular intervals (see item 1) as a function of their potential production rates (see “Well potentials” in the ECLIPSE Technical Description). For production wells, the drilling priorities are calculated from the formula

\[
\text{Priority} = \frac{A + B Q_o + C Q_w + D Q_g}{E + F Q_o + G Q_w + H Q_g} \tag{Eq. 3.63}
\]

where

- \( Q_o \), \( Q_w \) and \( Q_g \) are the well’s potential oil, water and gas production rates, and
- A-H are 8 coefficients set by the user with this keyword.

The equation allows the drilling priority to be set equal to, for example, the potential oil rate, or the reciprocal of the potential gas rate, or the reciprocal of the water cut. For injection wells, the drilling priorities are set equal to their potential injection rates. Individual wells may have their calculated priorities replaced by fixed values input in item 2 of keyword WDRILPRI, if required.

For injector wells this formula is not used and the drilling priority is calculated directly from the potential injection rate of the well unless an alternative drilling priority has been set using the WDRILPRI keyword.

The drilling priority of a well is based on the instantaneous production potential of that well unless items 10 and 11 are used. These items cause ECLIPSE to save the model state and run wells in the prioritized drilling queue for a period of time before calculating their priority. After these calculations the model is reset. This gives a better idea of the running state of the well. This “look-ahead” calculation of drilling priority is computationally expensive, so to avoid wasteful calculation priority it is only calculated when it is needed if item 1 is set to zero and item 10 is greater than zero.

The most recent calculation of well drilling priority may be output using WDRPR in the SUMMARY section. This will report zero for wells which are not in the prioritized drilling queue, either because they have not been added or because they have been drilled.

The DRILPRI keyword is followed by a line containing some or all of the following items of data:
1  The minimum time interval between drilling priority calculations.

Wells in the prioritized drilling queue have their drilling priorities calculated from the formula above at the beginning of each timestep that starts after the specified interval has elapsed since the previous calculation (except for wells given fixed drilling priorities in item 2 of keyword WDRILPRI).

A zero value causes the priorities to be recalculated every timestep if item 10 is defaulted. If item 10 is set, a zero value causes the drilling priority to be calculated when a drilling event is triggered.

UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

DEFAULT: Zero

2-9  The coefficients $A - H$ in the drilling priority formula.

None of the coefficients can have negative values. At least one of the first four coefficients ($A - D$) must be non-zero, and similarly at least one of the last four coefficients ($E - H$) must be non-zero.

DEFAULT: Zero

10  Lookahead period.

If this field is defaulted then drilling priorities are calculated using the potential rates of each well at the time of calculation. If this field is set to a positive value then ECLIPSE will save the model state, open one or more wells and run forward for this period before calculating the potentials to make the drilling priority calculation. The model is restored to the condition at the start of the drilling priority calculation once all well drilling priorities have been calculated.

If field 11 below is set to ‘SINGLE’ then the model state is run forward once for each well placed in the drilling queue by WDRILPRI, with one well opened and one drilling priority calculated per run forward. If field 11 is set to ‘ALL’ then all applicable wells are opened and have their drilling priority calculated at once. ‘SINGLE’ will give better results while ‘ALL’ will run more quickly as it involves fewer save, run, restore cycles.

The purpose of the look ahead calculation is to allow the engineer to determine the best available well if the initial flowing conditions are not likely to persist. This may be because water coning is likely to give a high water cut a short period after a well is opened, or it may be because an initial period of water production is expected from a coal bed methane development. If the look ahead period is set to a large value and the recalculation interval is set to a small interval then there will be significant performance implications.

UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

DEFAULT: Zero

11  Look ahead calculation type.

Available options for this field are ‘SINGLE’ and ‘ALL’. See field 10 for an explanation of this field.

DEFAULT: SINGLE

Examples

Example 1

Recalculate drilling priorities after every 100 days. Drill wells in decreasing order of oil potential.
Example 2
Recalculate drilling priorities at each timestep. Drill wells from lowest to highest water-liquid ratio.

```
DRILPRI
100  0.0  1.0  0.0  0.0  1.0  0.0  0.0  0.0  /
```

```
DRILPRI
  0  0.0  0.0  1.0  0.0  0.0  1.0  1.0  0.0  /
```
Maximum rate of increase of solution GOR

This keyword controls the rate at which the solution gas-oil ratio is allowed to rise. The keyword should be followed by some or all of the following items data terminated by a slash (/):

1. **DRSDT** Represents the maximum rate at which the solution gas-oil ratio in any grid block \( R_s \) is allowed to increase.
   
   **UNITS:** \( \text{sm}^3/\text{sm}^3/\text{day} \) (METRIC), Mscf/stb/day (FIELD), scc/scc/hr (LAB)

2. **Option flag**
   
   **ALL** Apply the **DRSDT** limit to all cells
   
   **FREE** Apply the **DRSDT** limit only to cells containing free gas

   **DEFAULT:** 'ALL'

This keyword controls the way in which free gas and undersaturated oil coexist within a grid block. For example, if **DRSDT** is set to 0, \( R_s \) cannot rise and free gas does not dissolve in undersaturated oil (no re-solution). At the other extreme, if **DRSDT** is very large, \( R_s \) rises very quickly until either the oil is saturated or no free gas remains (total re-solution).

If the keyword is not used, the resolution rate is unrestricted (equivalent to a resolution rate limit of infinity).

**Note:** This keyword has no effect on reductions in \( R_s \) (for example during pressure decline). Moreover increases in \( R_s \) are constrained by the availability of free gas and the oil saturation limit \( R_{ssat} \).

If this keyword is used in a system containing both dissolved gas and vaporized oil, the application of the **DRSDT** limit will be disabled by the subsequent use of the **VAPPARS** keyword. Similarly, the constraints imposed by the **VAPPARS** keyword will be disabled by the subsequent specification of the **DRSDT** keyword.

If \( \textbf{DRSDT} = 0 \) and the model contains undersaturated oil with a variation of \( R_s \) values, then non-physical behavior can occur when high \( R_s \) oil moves into a cell containing a lower \( R_s \). The **DRSDT** limit prevents the \( R_s \) increasing, and free gas develops, even though the oil is undersaturated. This behavior can be prevented by setting the option flag in item 2 to 'FREE'. The **DRSDT** limit then only applies to the two-phase regions. (Note that in this case gas flowing from a two-phase cell into an undersaturated cell will dissolve in the destination cell, as the \( R_s \) is allowed to rise.)

**Example**

```plaintext
DRSDT : 0.0003 / (in Mscf/stb/day)
```
Radial-direction grid block sizes (vector)

This keyword specifies the size of the cells in the radial direction.

The keyword should be followed by up to NDIVIX positive real numbers, where NDIVIX is the grid radial dimension entered using the DIMENS keyword in the RUNSPEC section.

The I\textsuperscript{th} number specifies the size in the radial direction of all the grid blocks with the R axis index equal to I. In other words, blocks (I, 1-NDIVIY, 1-NDIVIZ).

DRV causes the DR value for every grid block in the reservoir to be set. Repeat counts may be used for repeated values (for example 11*208.4). Note that spaces must not be inserted on either side of the asterisk.

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

The data must be terminated by a slash (/).

Notes

- The inner radius of the innermost column (I = 1) of grid blocks must be set. To do this, refer to the INRAD keyword, for ECLIPSE 100, and the RADFIN keyword, for ECLIPSE 300. These blocks have an outer radius equal to the sum of the inner radius and their DRV value.

- If the outer radius is not entered (either using the OUTRAD keyword in ECLIPSE 100, or the RADFIN keyword in ECLIPSE 300), a complete set (NDIVIX) of DRV values must be entered. The outer radius of these blocks is equal to the sum of the inner radius and the accumulated DRV values.

- If keyword OUTRAD is entered, it is not necessary to specify all (or any) DRV values. For ECLIPSE 100, any DRV values not specified are completed using a geometric series (see keyword OUTRAD), and for ECLIPSE 300, any DRV values not specified are completed using a logarithmic series.

- This keyword cannot be used to specify the grid data for a Local Grid Refinement (between the RADFIN and ENDFIN pair).

Example

With NDIVIX=11 as specified in RUNSPEC keyword DIMENS:

```plaintext
DRV
1 2 4 8 16 32 64 128 256 512 1024 /
INRAD
1 /
```
Maximum rate of increase of vapor OGR

The keyword should be followed by a single real number which represents the maximum rate at which the vapor oil-gas ratio in any grid block (RV) is allowed to increase. The data field should be terminated by a slash (/).

This keyword controls the way in which free oil and undersaturated gas coexist within a grid block. For example, if DRVDT is set to 0, RV cannot rise and free oil does not vaporize in undersaturated gas (no re-vaporization). At the other extreme, if DRVDT is very large, RV rises very quickly until either the gas is saturated or no free oil remains (total re-vaporization).

If the keyword is not used, the re-vaporization rate is unrestricted (equivalent to a re-vaporization rate limit of infinity).

**Note:** This keyword has no effect on reductions in RV.

Moreover increases in RV are constrained by the availability of free oil and the gas saturation limit (Rvsat).

If this keyword is used in a system containing both dissolved gas and vaporized oil, the application of the DRVDT limit will be disabled by the subsequent use of the VAPPARS keyword. Similarly, the constraints imposed by the VAPPARS keyword will be disabled by the subsequent specification of the DRVDT keyword.

**Example**

```
DRVDT
0.009 /
```
Interpolation of dispersion tables by water density

When using the tracer dispersion option for a BRINE tracer, item 2 of the DISPERSE keyword can be changed from tracer concentration to water density by using this keyword.

See also keywords DISPDIMS, TRDIS and DISPERSE.

The DSPDEINT keyword has no associated data.

Example

```
DSPDEINT
```

Keywords

ECLIPSE Reference Manual
**DSTNUM**  Tertiary hysteresis saturation function region numbers

The keyword should be followed by one integer for every grid block in the current input box specifying the tertiary hysteresis saturation function region to which it belongs. This keyword is used together with **PSTNUM** and **ISTNUM** if the **ODD3P** relative permeability and capillary pressure hysteresis model has been selected in the **PROPS** section. The region number should not be less than 1 or greater than **NTSFUN** (see keyword **TABDIMS**). The data must be terminated by a slash (/).

The tertiary hysteresis saturation function region number specifies which set of saturation functions (input using **SOF3**, **SGF3**, and **SWF3** in the **PROPS** section) should be used to calculate the tertiary hysteresis relative permeabilities and capillary pressures in each grid block.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

For grid blocks for which the tertiary hysteresis saturation function region numbers are not specified, the values will be assigned from those defined for the corresponding secondary hysteresis saturation function region numbers using the **ISTNUM** keyword. The **ODD3P** model is implicitly hysteretic. However, hysteresis may be deselected on a per grid block basis by setting the secondary and tertiary hysteresis saturation region numbers equal to the primary saturation region numbers.

See also the saturation function keywords **SOF3**, **SGF3** and **SWF3** in the **PROPS** section and the primary and secondary hysteresis saturation region numbers **PSTNUM** and **ISTNUM** in the **REGIONS** section.

**Example**

With **NTSFUN=2**, **NDIVIX=8**, **NDIVIY=6** and **NDIVIZ=5**, as specified in **RUNSPEC** keyword **DIMENS**, and no input **BOX** set:

```
DSTNUM
144*1 96*2 /
```

Keywords 668
DTHETA

Theta-direction grid block sizes

The keyword should be followed by one positive real number for every grid block in the current input box specifying its size in the theta direction. The data must be terminated by a slash (/).

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta and Z axis indices. Repeat counts may be used for repeated values (for example 115*72). Note that spaces must not be inserted on either side of the asterisk.

UNITS: degrees (METRIC), degrees (FIELD), degrees (LAB), degrees (PVT-M)

In ECLIPSE 100, every DTHETA value in the top plane (K = 1) must be specified in one way or another by the end of the GRID section. Values in lower planes (K > 1) which are not specified default to the value in the plane above.

Example

```
BOX   1  12  1  6  1  1  /
DTHETA
6*60 59 58 57 55 53 50
6*60  61 62 63 65 67 70
12*40
12*53
12*67
12*80  /
```

Keywords

ECLIPSE Reference Manual

669
Angular sizes of grid blocks (vector)

This keyword specifies the size of the cells in the theta direction.

The keyword should be followed by DIVIY positive real numbers, where NDIVIY is the grid theta dimension entered using the DIMENS keyword in the RUNSPEC section.

The J-th number specifies the angular size in the theta direction of all the grid blocks with the theta axis index equal to J. In other words, blocks (1-NDIVIX, J, 1-NDIVIZ).

DTHETAV causes the DTHETA value for every grid block in the reservoir to be set. Repeat counts may be used for repeated values (for example 6*60). Note that spaces must not be inserted on either side of the asterisk.

UNITS: degrees (METRIC), degrees (FIELD), degrees (LAB), degrees (PVT-M)

The data must be terminated by a slash (/).

Note: In ECLIPSE 300, this keyword must not be used to specify the grid data for a Local Grid Refinement (between the RADFIN and ENDFIN pair).

Example

With NDIVIY=8, specified in RUNSPEC keyword DIMENS:

```
  DTHETA
  20 20 30 40 4*50 / entire reservoir subtends 310 degrees
```
DUALPERM  Run is to use dual permeability

This requests that the dual permeability option is used in the run.

This extends the dual porosity option to allow flow directly between matrix cells. It is not necessary to specify the DUALPORO keyword as well as DUALPERM, as the DUALPERM keyword will also enable the dual porosity option.

See "Dual porosity" in the ECLIPSE Technical Description for further information.

The keyword has no associated data.

Note: A faster linear solver was introduced in 97A for the solution of dual permeability problems. With some problems the pre-97A solver may prove to be more robust: it can be turned on by using item 60 of the OPTIONS keyword in RUNSPEC.
DUALPORO  Run is to use dual porosity

This requests that the dual porosity option is used in the run.

If either DUALPORO or dual permeability (DUALPERM) are used, the number of layers entered in item 3 of the DIMENS keyword must be even. ECLIPSE interprets the first half of the grid layers as matrix cells, and the remainder as fracture cells. The non-neighbor connections representing matrix-fracture flow transmissibilities are automatically constructed.

See "Dual porosity" in the ECLIPSE Technical Description for further information.

The keyword has no associated data.
The **DUMPCUPL** keyword may be used in a Reservoir Coupling master run if you wish it to write the flows and applied constraints of the master groups to the output Reservoir Coupling file (see "Reservoir coupling files" in the *ECLIPSE Technical Description*).

The file may then be used for two purposes:

- In the master run, it can take the place of one or more of the slave reservoirs. Their slave runs are not activated, and instead the associated master groups take their production and injection rates from this file.
- A slave run may be performed on its own, instead of being activated by the master run. It will read from this file the constraints that the master run originally applied to its slave groups over the course of the simulation, and apply them again in the same sequence.

Runs using the Reservoir Coupling file must have the keyword **USECUPL** entered.

The **DUMPCUPL** keyword is followed by a record containing a single character, terminated with a slash (/). The character dictates the type of file to be written. The choices are:

- **U**: Write an unformatted file
- **F**: Write a formatted file.

Formatted files occupy much more disk space than unformatted files, but should be chosen if the Reservoir Coupling file is going to be used on a system that does not have binary compatibility with the one on which it was created. Note that the formatted/unformatted status of the Reservoir Coupling file can be chosen independently of keyword **FMTOUT** in the RUNSPEC section.

The Reservoir Coupling file created has the same root name as the Master Run’s data file. The filename’s extension is **.CPL** for an unformatted file, and **.FCP** for a formatted file.

Note that the **DUMPCUPL** instruction is **not** carried over on the Restart file. If you wish to restart the Master Run and continue writing a Reservoir Coupling file (which is a new file having the restart run’s root name), you must include the **DUMPCUPL** keyword in the restart run’s SCHEDULE section. A suitable place for the keyword is at the beginning of the SCHEDULE section, or next to the **SLAVES** keyword.

### Example

```
DUMPCUPL
U /
```
DUMPFLUX  Makes a full-field run write a flux file

The DUMPFLUX keyword is used in the full-field run to tell ECLIPSE to write a Flux file containing the flows across the flux region boundaries.

If the base run is either a Blackoil or a Compositional model, and the reduced run is a Thermal model, then the keyword FLUXOPTS must be used to tell ECLIPSE to output the necessary information for the reduced case. For a full field thermal to reduced thermal case, the FLUXOPTS keyword is not required.

The extent of the flux regions are defined by the FLUXNUM keyword.

The keyword has no associated data.

See "Flux boundary conditions" in the ECLIPSE Technical Description for further information about this option.

Note: A thermal reduced run can only be carried out from an ECLIPSE 300 base run.
**X-direction grid block sizes**

This keyword specifies the size of the cells in the X-direction.

The keyword should be followed by one positive real number for every grid block in the current input box specifying its size in the X-direction.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

The data must be terminated by a slash (/).

**Examples**

**Example 1**

```
BOX 5 16 3 8 1 1 /
DX 1000 500 10*600
    900 600 10*600
    800 700 10*650
    700 800 10*700
    600 900 10*750
    500 1000 10*750 /
```

**Example 2**

- Two-layer grid 10 x 10 x 2 grid
- 200 ft by 200 ft by 10 ft cells in top layer
- 200 ft by 200 ft by 50 ft cells in bottom layer

```
DX 200*200 /
```

**ECLIPSE 100**

Every DX value in the top plane (K = 1) must be specified in some way by the end of the GRID section. Values in lower planes (K > 1) which are not specified default to the value in the plane above.
DXV

X-direction grid block sizes (vector)

This keyword specifies the size of the cells in the X-direction.

The keyword should be followed by NDIVIX positive real numbers, where NDIVIX is the grid X dimension entered using the DIMENS keyword in the RUNSPEC section.

The I\textsuperscript{th} number specifies the size in the X-direction of all the grid blocks with the X axis index equal to I. In other words, blocks (I, 1-NDIVIY, 1-NDIVIZ). See the DIMENS keyword for the specification of NDIVIY and NDIVIZ.

DXV causes the DX value for every grid block in the reservoir to be set. Repeat counts may be used for repeated values (for example 11*208.4). Note that spaces must not be inserted on either side of the asterisk.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

The data must be terminated by a slash (/).

**Note:** This keyword must not be used to specify the grid data for a Local Grid Refinement (between the CARFIN and ENDFIN pair).

**Example**

With NDIVIX = 11, specified in RUNSPEC keyword DIMENS:

```
DXV
1000 500 7*200 500 1000 /
```
DY

### Y-direction grid block sizes

This keyword specifies the size of the cells in the Y-direction.

The keyword should be followed by one positive real number for every grid block in the current input box specifying its size in the Y-direction.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

**UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

The data must be terminated by a slash (/).

#### Examples

**Example 1**

```
----- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX   5 16 3 8 1 1 /
DY
1000 950 900 850 800 750 700 650 600 550 500 450
650 700 750 800 850 900 950 1000 1050 1100 1150 1200
48*2020.9 /
```

**Example 2**

- Two-layer grid 10 x 10 x 2 grid
- 200 ft by 200 ft by 10 ft cells in top layer
- 200 ft by 200 ft by 50 ft cells in bottom layer

```
DY 200*200 /
```

**ECLIPSE 100**

Every **DY** value in the top plane (**K = 1**) must be specified in some way by the end of the **GRID** section. Values in lower planes (**K > 1**) that are not specified default to the value in the plane above.

---

Keywords

677
**DYNAMICR**  

**Define regions based on when a set of conditions are satisfied**

This keyword may be used to define regions based on when a set of conditions are satisfied, with conditions defined using Boolean logic and collections of objects. These regions may be static or allowed to change dynamically. These new regions can be used with ECLIPSE keywords, for instance ACTIONX and **UDQ**.

This will allow property and reporting regions to be defined and altered throughout the run by user definition.

The keyword is followed by a series of records each terminated by a slash (/) with the keyword terminated by a blank record containing only a slash (/):

The data items in each record are:

**Record 1**

1. The region name.
   
   A name of up to 8 characters, to distinguish this region name from any others. The first 3 characters must be “FIP”. The family name is a string of up to 5 characters that makes up the second part of the keyword name (characters 4-8). These names must be distinct from ordinary FIP region families defined in the REGIONS section.

2. Region number.
   
   This will be the region number affected by the conditions specified in subsequent records. The value specified here should be an integer between 0 and MNUMDR, given in item 1 of DYNRDIMS keyword.

   **DEFAULT:** 0

**Each subsequent record**

The conditions defined will be applied to the region number and name defined in record 1 above.

Please refer to "Each subsequent record" for details of the format of a condition and lists of mnemonics.

When DYNAMICR is specified in the SOLUTION section, many of the mnemonic values are not meaningful - for instance, no wells have yet been defined. Conditions containing undefined values will be ignored and a warning given.

**Specific requirements for DYNAMICR conditions**

The left hand side of the condition must be either a block keyword or one of the DYNAMICR special keywords. Grid indexes are not specified for the block keywords as each grid block will be considered individually in turn.

To create a DYNAMICR condition which is also dependent on a condition which is not evaluated block-by-block, the non-block-by-block condition should be specified in an ACTIONX keyword so that the DYNAMICR keyword is only evaluated if the other condition is fulfilled. See examples.

**DYNAMICR special mnemonics**

Note that these mnemonics cannot be used with ACTIONX, and that the double quotes are required as part of the mnemonic.
DIST_TO_GRIDBLOCK  This can be used on either the left or right hand side of a condition - or both. It
is followed by the x, y, z indexes of the reference grid block. The distance to
the center of this grid block is then evaluated for each grid block.

One or more of the indexes can be defaulted. If this is done, the default value
used for each evaluation will be the same as that index for the grid block to
which the distance is being calculated. This allows the calculation of the
distance to a row or layer of cells rather than a single cell.

Note: Indexes cannot be defaulted when running in parallel.

DIST_TO_WELLNAME  This can be used on either the left or right hand side of a condition - or both. It
is followed by the name of the well and the number of the connection to be
used. The distance to the center of the grid block in which the connection is
situated is then evaluated for each grid block. Note that the connection number
refers to a single grid block only. This mnemonic cannot be used in the
SOLUTION section since the wells have not yet been defined. See Example 5.

Template characters or well lists can be used in place of a well name. However, only the first well which
matches the requirements will be considered. If, for instance, the minimum distance to one of two wells is
required, two separate conditions on the two wells must be specified and combined using OR.

DYNAMICR special conditions

Note that these conditions cannot be used with ACTIONX, and that the double quotes are required as part of
the condition.

When “region families” are referred to generically, they can be either dynamic region families, or ordinary
FIP region families defined in the REGIONS section.

ALL    this can be used in two ways:

• As an entire condition by itself. This allows a default region value to be set for a particular
dynamic region family without having to evaluate a quantity for every grid block.

• Or by following it with pairs of region family name + region number. This allows a
dynamic region to be set up which combines all the cells in several regions. Up to three
pairs can be specified per condition; if more than this are required then multiple ALL
conditions can be combined using the standard methods for combining conditions.

IN     this condition has no associated value on the left hand side. The right hand side has a region
family name and region number. Each grid block’s region number is set to the specified value if
it is in the given region. Used alone, it simply duplicates an existing region. Its utility comes
when it is combined with other conditions.

NOT_IN this condition has no associated value on the left hand side. The right hand side has a region
family name and region number. Each grid block’s region number is set to the specified value if
it is not in the given region. It can be used simply to set up a region consisting of all those grid
blocks not in an existing region, or in combination with other conditions.
Note: While it is possible to use NOT_IN to set a default region value for all cells which don’t meet an already calculated complex condition, it requires much less computation to use ALL to set a default value for all cells first.

Local grids

Dynamic regions cannot be specified for local grid blocks. However, if required, local grid mnemonics can be used on the right hand side of a condition in exactly the same way as for ACTIONX.

The methods of use for this keyword are:

Statically

DYNAMCR is used to define a region family in the SOLUTION section, or only in the SCHEDULE section. Once defined, the region values for the cells never change. This is very similar to an ordinary FIP region family defined in the REGIONS section, except that conditions can be used to determine the region values. See Example 1.

Dynamically - automatic

The DYNAMCR keyword is specified inside an ACTIONX keyword using the ALWAYS condition. Region values will be recomputed every timestep. See Example 2.

Dynamically - conditional

The DYNAMCR keyword is specified inside an ACTIONX keyword using any condition. Normal ACTIONX functionality is used to determine when the region values will be recomputed. See ACTIONX examples for sample conditions.

Dynamically - manual

The DYNAMCR keyword is specified at multiple points within the SCHEDULE section for the same dynamic region family. Region values will be recomputed each time the keyword is encountered. There is no reason why the conditions on the region values should be the same in all definitions. See Example 3.

These methods can be freely combined as required for a single dynamic region family.

Notes

- The order of the items in the quantities is similar to the way summary mnemonics are defined, that is the mnemonic is the first item and it is then followed by a well or group name for example.

- No checks are in place on the type of quantities that are compared in a condition. So for example, a condition on this distance between two grid blocks being less than a connection’s water cut would be allowed.

- Unlike ACTIONX, details of whether conditions are met are not written to file - since they are calculated for every block.

- Full checking of the keywords entered with a DYNAMCR keyword cannot be performed until the action is actually implemented, since it is possible to use wild cards in many of the keywords. However, before a simulation run is attempted the data can be run in NOSIM mode. This then does whatever checks it can on the keywords within any keyword blocks that have been defined. All the most common types of ‘grammatical’ and typographical errors will be found, but errors resulting from wild cards feeding inappropriate well/group names to keywords are not found.

- When using DYNAMCR inside an ACTIONX keyword, it is important to note that no keywords inside an ACTIONX keyword are processed until the following timestep is run. This means that, for instance, it is not possible to specify a dynamic region in an ACTIONR keyword immediately following the ACTIONX keyword in which it is defined.
• The DYNREG argument has been added to the RPTRST keyword in both the SOLUTION and SCHEDULE sections. This outputs the cell indices corresponding to the dynamic regions specified using the DYNAMICR keyword.

• When running in NOSIM mode, the block-by-block calculations of region numbers are not performed for speed reasons, so values output to the restart file with the DYNREG argument are not meaningful.

Examples

Only simple right hand conditions have been used here to demonstrate the use of the new DYNAMICR facilities. For examples of more complex conditions, please see the ACTIONX examples.

Example 1

This sets up a dynamic region family FIPYYY for which region 1 is initially identical to region 1 of the region family FIPXXX. Region 2 contains all those cells which are in either FIPXXX regions 2 or 3. Region 3 consists of all those grid blocks with block oil in place less than or equal to 500,000. Note that any cells which fulfil none of the conditions will be in region 1 by default. This dynamic region will not be updated.

```
DYNAMICR
  FIPYYY 1 /
    "IN" FIPXXX 1 /
  FIPYYY 2 /
    "ALL" FIPXXX 2 FIPXXX 3 /
  FIPYYY 3 /
    BOIP <= 500000 /
/ENDDYN
```

Example 2

The dynamic region family FIPYYY is set up exactly as in the previous example. However, due to being wrapped in an ACTIONX keyword with the ALWAYS condition, the region numbers for each cell will be recalculated every timestep.

```
ACTIONX
  ACT2 10000 /
    "ALWAYS"/
/ENDACTIO
DYNAMICR
  FIPYYY 1 /
    "IN" FIPXXX 1 /
  FIPYYY 2 /
    "ALL" FIPXXX 2 FIPXXX 3 /
  FIPYYY 3 /
    BOIP <= 500000 /
/ENDDYN
```

Example 3

This sets up the same dynamic region family FIPYYY as example 1. Then, after a time interval of 10, all grid blocks with BOIP less than 400000 are added to region 3.

```
ENDDYN
```
Note that no other grid blocks will have their region values changed at all - any grid block which was previously part of region 3 will remain part of it, even if its BOIP value is now greater than 400,000. To remove cells which no longer fit the condition, the second DYNAMICR keyword would have to also recompute regions 1 and 2.

DYNAMICR
  FIPYYY  1 /   "IN" FIPXXX 1 /
  FIPYYY  2 /   "ALL" FIPXXX 2 FIPXXX 3 /
  FIPYYY  3 /   BOIP <= 500000 /
  ENDDYN

TSTEP
  10 /

DYNAMICR
  FIPYYY  3 /
  FIPYYY  2 /
  ENDDYN

Example 4

These two uses of the DYNAMICR keyword produce exactly the same region values, but the first one is to be greatly preferred as only half as many distance calculations are required.

Dynamic region family FIPYYY is set up so that all grid blocks are in region 2 except for those which are closer to grid block 3,3,2 than to grid block 5,5,1.

DYNAMICR
  FIPYYY  2/   "ALL" /
  FIPYYY  1/   "DIST_TO_GRIDBLOCK" 3 3 2 <= "DIST_TO_GRIDBLOCK" 5 5 1/
  ENDDYN

DYNAMICR
  FIPYYY  1/   "DIST_TO_GRIDBLOCK" 3 3 2 <= "DIST_TO_GRIDBLOCK" 5 5 1/
  FIPYYY  2/   "DIST_TO_GRIDBLOCK" 3 3 2 > "DIST_TO_GRIDBLOCK" 5 5 1/
  ENDDYN

Example 5

This demonstrates the use of the DIST_TO_WELLNAME mnemonic.

Dynamic region family FIPYYY is set up so that all grid blocks are in region 2 except for those which are within 10 of the second connection of well INJ.

This will give exactly the same results as using the DIST_TO_GRIDBLOCK mnemonic with the x,y,z indexes of connection 2.

DYNAMICR
  FIPYYY  2/   "ALL" /
  FIPYYY  2/   "DIST_TO_WELLNAME" INJ 10 /
  ENDDYN
FIPYYY 1/
   "DIST_TO_WELLNAME" INJ 2 <= 10 /

ENDDYN
DYNRDIMS Specify dimensions required for the dynamic region feature

This keyword determines the size of storage set aside for dynamic regions specified with the DYNAMICR keyword, and must be present if dynamic regions are to be specified.

The data consists of some or all of the following items.

The data must be terminated by a slash (/).

1. The maximum number of dynamic regions (MNUMDR).

   If item 1 of REGDIMS or item 5 of TABDIMS are also set, the greatest value of the three settings is used.

   **Note:** Since the fluid-in-place balance sheet reports region-to-region flows, the storage required is proportional to the square of MNUMDR. Very large values of MNUMDR should therefore be avoided, as excessive storage may be required and in addition the restart files will be very large.

   DEFAULT: 0

2. The maximum number of dynamic region families (MXDYNF).

   DEFAULT: 0

3. The maximum number of dynamic region records which can be defined under the DYNAMICR keyword (MXDYNR).

   DEFAULT: 0

**Example**

```
DYNRDIMS 5 2 10 /
```
DYV

Y-direction grid block sizes (vector)

This keyword specifies the size of the cells in the Y-direction.

The keyword should be followed by NDIVY positive real numbers, where NDIVY is the grid Y dimension entered using the DIMENS keyword in the RUNSPEC section.

The J\textsuperscript{th} number specifies the size in the Y-direction of all the grid blocks with the Y axis index equal to J. In other words, blocks (1-NDIVIX, J, 1-NDIVIZ). See the DIMENS keyword for the specification of NDIVIX and NDIVIZ.

DYV causes the DY value for every grid block in the reservoir to be set. Repeat counts may be used for repeated values (for example 11*208.4). Note that spaces must not be inserted on either side of the asterisk.

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

The data must be terminated by a slash (/).

\textbf{Example}

With NDIVY = 11, specified in RUNSPEC keyword DIMENS:

```
DYV
1000 500 7*200 500 1000 /
```
**Z-direction grid block sizes**

This keyword specifies the size of the cells in the Z-direction.

The keyword should be followed by one positive real number for every grid block in the current input box specifying its size in the Z-direction.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

**UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

The data must be terminated by a slash (/).

**Notes**

**ECLIPSE 100**

Every **DZ** value in the top plane (K = 1) must be specified in one way or another by the end of the **GRID** section. Values in lower planes (K > 1) that are not specified default to the value in the plane above.

**ECLIPSE 300**

If **DZ** <= 10^-6 for any cell, the cell will be either be made inactive or in thermal cases it may become rock-filled.

**Examples**

**Example 1**

```
BOX
5  16   3   8   1   1   /
DZ
20 25 25 23 21 18 16 15 15 14 14 14
20 25 25 23 21 18 16 15 15 14 14 14
20 25 25 23 21 18 16 15 15 14 14 14
20 25 25 23 21 18 16 15 15 14 14 14
20 25 25 23 21 18 16 15 15 14 14 14 /
```

**Example 2**

- Two-layer grid 10 x 10 x 2 grid
- 200 ft by 200 ft by 10 ft cells in top layer
- 200 ft by 200 ft by 50 ft cells in bottom layer

```dzone
DZ
100*10 100*50 /
```
DZMATRIX  The vertical dimension of a block of matrix material

This is an alias for DZMTRXV.
DZMTRX  The vertical dimension of a block of matrix material

If the dual porosity gravity imbibition option is being used (keywords DUALPORO and GRAVDR or GRAVDRM, or the VERTICAL option of NMATOPTS, in the RUNSPEC section), this keyword should be used to specify the vertical dimension of a typical block of matrix material (see also $l_v$ in keyword SIGMA).

The strength of the gravity imbibition effect is directly proportional to DZMTRX. The effect is usually very small. However, in the absence of capillary pressure, there is no other mechanism to equalize the fluid levels in matrix and corresponding fracture cells. For further details, see "Dual porosity" in the ECLIPSE Technical Description.

This keyword specifies the DZMTRX factor for the entire grid. However the DZMTRX factor can be specified on a cell by cell basis using the DZMTRXV keyword.

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: 0.0 (Throughout the grid.)

Note: This is typically much smaller than the vertical dimension of a simulation matrix cell, DZ.

Example

In the following example, the vertical dimension of a block of matrix material is 2ft.

```
DZMTRX
  2.0 /
```
**DZMTRXV**  

The vertical dimension of a block of matrix material

If the dual porosity gravity imbibition option is being used (keywords DUALPORO and GRAVDR, GRAVDRM, or the VERTICAL option of NMATOPTS, in the RUNSPEC section), either this keyword or keyword DZMTRX should be used to specify the vertical dimension of a typical block of matrix material. This keyword allows the DZMTRX values to be specified on a cell by cell basis.

The strength of the gravity imbibition effect is directly proportional to DZMTRX. The effect is usually very small. However, in the absence of capillary pressure, there is no other mechanism to equalize the fluid levels in matrix and corresponding fracture cells. For further details, see “Dual porosity” in the ECLIPSE Technical Description.

The keyword should be followed by one positive real number for every grid block in the current box within the top NDIVIZ /2 layers of the grid. Any values input in the bottom half of the grid (the lower NDIVIZ /2 layers) are ignored. When the DZMTRX values are output (mnemonic DZMTRXV in the RPTGRID keyword) the first NDIVIZ /2 layers are copied into the lower half of the grid.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: 0.0 (throughout the grid.)

**Note:** This is typically much smaller than the vertical dimension of a simulation matrix cell, DZ.

---

**Example**

In the following example, the vertical dimension of a block of matrix material is 2ft.

```plaintext
**** IX1-IX2 JY1-JY2 KZ1-KZ2
BOX          1 4 1 2 1 1 /
DZMTRXV      8*2.0 /
```
**DZNET**

### Specifies the cell net DZ values

This keyword specifies the net thickness value for each grid block. These values are used in the calculation of grid block pore volumes and transmissibilities in the X and Y (or R and theta) directions.

If **DZNET** is used, the net to gross ratio is calculated as follows:

\[
NTG_i = \frac{DZNET_i}{DZ_i}
\]  

Eq. 3.64

**UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

Any **DZNET** values which are not specified when the end of the GRID section is reached default to the gross grid block thickness (entered using **DZ**).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

The keyword should be followed by one non-negative real number for every grid block in the current input box.

The data must be terminated with a slash (/).

---

**CAUTION:** The use of both **DZNET** and **NTG** keywords in the same run is not allowed.

---

### Example

```plaintext
-- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
6 11 4 9 2 3 /
DZNET
6*8 6*8 6*9.6 6*10.2 6*14 6*14.2
6*8 6*8 6*9.6 6*10.2 6*14 6*14.2 /
```

---

### Keywords

- **ECLIPSE 100**
- **ECLIPSE 300**
- **SPECIAL**
- **RUNSPEC**
- **GRID**
- **EDIT**
- **PROPS**
- **REGIONS**
- **SOLUTION**
- **SUMMARY**
- **SCHEDULE**
**DZV**

### Z-direction grid block sizes (vector)

This keyword specifies the size of the cells in the Z-direction.

The keyword should be followed by `NDIVIZ` positive real numbers, where `NDIVIZ` is the grid Z dimension entered using the `DIMENS` keyword in the `RUNSPEC` section.

The `Kth` number specifies the size in the Z-direction of all the grid blocks with a Z axis index equal to `K`. In other words, blocks `(1-NDIVIX,1-NDIVIY,K)`. See the `DIMENS` keyword for the specification of `NDIVIX` and `NDIVIY`.

`DZV` causes the `DZ` value for every grid block in the reservoir to be set. Repeat counts may be used for repeated values (for example `11*208.4`). Note that spaces must not be inserted on either side of the asterisk.

**UNITS:** m (`METRIC`), ft (`FIELD`), cm (`LAB`), m (`PVT-M`)

The data must be terminated by a slash (`/`).

### Notes

This keyword **must not** be used to specify the grid data for a Local Grid Refinement (between the `CARFIN` / `RADFIN` and `ENDFIN` pair).

If `DZ <= 10^-6` for any cell, the cell will be either be made inactive or in thermal cases it may become rock-filled.

### Example

```
DZV
140 150 3*155 160 190 200 191 204 4*300 /
```
This section contains an alphabetic listing of keywords starting with the letter E.
Enable NOSIM if incompatible E100 keyword

This keyword causes a switch to NOSIM mode if any ECLIPSE 100 keywords are encountered that are not recognized or are substantially different from their ECLIPSE 300 counterparts.

The simulator will also switch to NOSIM mode if it reads a non-default entry in an item that is not supported by ECLIPSE 300 in a keyword that is recognized by both simulators.
ECHO

Switches echo output on

The keyword turns on the echoing of the input file to the Print file at the start of the run. This echoing may be disabled using NOECHO, for example when large machine generated input files are entered.

The default status for echoing is on, which may be set to off using NOECHO. ECHO is normally only required, therefore, to turn echoing back on after a NOECHO keyword.

ECHO and NOECHO may be specified in any section, and any number of times in an input file.
ECLMC

Activates multi-component option

The keyword activates the multi-component option in ECLIPSE 100. The keyword is required in runs using the multi-component brine model.

Note that the multi-component brine model is distinct from the standard brine option (latter specified using keyword `BRINE` without keyword `ECLMC`). Thus, functionality that is specific to the standard brine model, such as `PLYVISCS`, is not compatible with `ECLMC`.

The keyword has no associated data.

See "Brine tracking" in the *ECLIPSE Technical Description*. 

---

### Keywords

<table>
<thead>
<tr>
<th>x</th>
<th>ECLIPSE 100</th>
<th>ECLIPSE 300</th>
<th>SPECIAL</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>RUNSPEC</td>
<td>GRID</td>
<td>EDIT</td>
</tr>
<tr>
<td></td>
<td>PROPS</td>
<td>REGIONS</td>
<td>SOLUTION</td>
</tr>
<tr>
<td></td>
<td>SUMMARY</td>
<td>SCHEDULE</td>
<td></td>
</tr>
</tbody>
</table>
Multiply a non-neighbor connection

Non-neighbor connections may be generated by geological faults in the grid. This keyword can be used to modify these fault generated non-neighbor connections.

Each line following the EDITNNC keyword specifies a non-neighbor connection to be modified and is terminated with a slash (/). After the last non-neighbor modification, a single (/) terminates the list.

The items in each line are:

1. I-index of first cell joined to the non-neighbor connection (IX)
2. J-index of first cell joined to the non-neighbor connection (IY)
3. K-index of first cell joined to the non-neighbor connection (IZ)
4. I-index of second cell joined to the non-neighbor connection (JX)
5. J-index of second cell joined to the non-neighbor connection (JY)
6. K-index of second cell joined to the non-neighbor connection (JZ)

The cell coordinates must be defined and cannot be defaulted.

7. Transmissibility multiplier for the non-neighbor connection (TRANM).
   The transmissibility is set to TR*TRANM, where TR is the non-neighbor connection transmissibility calculated by the program or entered directly using the NNC keyword.
   The multiplier TRANM should not be negative but can be zero.
   DEFAULT: 1.0

8. Saturation table number to be associated with flow from the first to the second cell (IST1)
   DEFAULT: 0

9. Saturation table number to be associated with flow from the second to the first cell (IST2)
   DEFAULT: 0

10. Pressure table number to be associated with flow from the first to the second cell (PT1)
    DEFAULT: 0

11. Pressure table number to be associated with flow from the second to the first cell (PT2)
    DEFAULT: 0

12. The face associated with flow from the first to the second cell (ZF1)
    Choose either: X+, X-, Y+, Y-, Z+, Z-
    This information is used with the VE option only

13. The face associated with flow from the second to the first cell (ZF2).
    Choose either: X+, X-, Y+, Y-, Z+, Z-
    This information is used with the VE option only

14. Diffusivity multiplier for the non-neighbor connection (DIFFM)

ECLIPSE 100 only

ECLIPSE 300

SPECIAL

RUNSPEC

GRID

EDIT

PROPS

REGIONS

SOLUTION

SUMMARY

SCHEDULE

Keywords

696
The diffusivity is set to $DF \times DIFFM$, where $DF$ is the non-neighbor connection diffusivity calculated by the program or entered directly using the NNC keyword.

The multiplier $DIFFM$ should not be negative but can be zero.

DEFAULT: 0

The last seven items are optional: if they are input then they supersede the original table numbers and VE flow faces. However if a slash (/) is placed after the TRANM item, then the original table numbers and VE flow faces will remain and diffusivities will not be altered.

**Note:** When the Local Grid Refinement option is employed, the EDITNNC keyword cannot be used within local grids. If an NNC is edited in the global grid, then the corresponding NNCs in any local grids are **not** edited. In this case, it is advisable to employ multipliers, for example MULTX, instead (where applicable) or else consider using inter-region transmissibility multipliers defined by keyword MULTREGT to replace the EDITNNC records.

If a non-neighbor connection to be edited cannot be found, then a warning message is issued and the EDITNNC line ignored.

**Example**

```
EDITNNC
  IX IY IZ JX JY JZ TRANM IST1 IST2 IPT1 IPT2 /
  .
  .
/
```

**Keywords**
**EDITNNCR**

**Replace a non-neighbor connection**

Non-neighbor connections may be generated by geological faults in the grid. This keyword can be used to replace these fault-generated non-neighbor connections.

Each line following the **EDITNNCR** keyword specifies a non-neighbor connection to be replaced and is terminated with a slash (/). After the last non-neighbor replacement, a single slash (/) terminates the list.

The items in each line are:

1. I-index of first cell joined to the non-neighbor connection (IX)
2. J-index of first cell joined to the non-neighbor connection (IY)
3. K-index of first cell joined to the non-neighbor connection (IZ)
4. I-index of second cell joined to the non-neighbor connection (JX)
5. J-index of second cell joined to the non-neighbor connection (JY)
6. K-index of second cell joined to the non-neighbor connection (JZ)

   The cell coordinates must be defined and cannot be defaulted.

7. Transmissibility for the non-neighbor connection (TRANS)
8. Saturation table number to be associated with flow from the first to the second cell (IST1)
   
   DEFAULT: value calculated in GRID section

9. Saturation table number to be associated with flow from the second to the first cell (IST2)
   
   DEFAULT: 0

10. Pressure table number to be associated with flow from the first to the second cell (IPT1)
    
    DEFAULT: 0

11. Pressure table number to be associated with flow from the second to the first cell (IPT2)
    
    DEFAULT: 0

12. The face associated with flow from the first to the second cell (ZF1)
    
    Choose either: X+, X-, Y+, Y-, Z+, Z-
    
    This information is used with the VE option only

13. The face associated with flow from the second to the first cell (ZF2)
    
    Choose either: X+, X-, Y+, Y-, Z+, Z-
    
    This information is used with the VE option only


    UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

    DEFAULT: Value calculated in GRID section.

**ECLIPSE 100**

The last seven items are optional: if they are input then they supersede the original table numbers and VE flow faces. However if a slash (/) is placed after the TRANS item, then the original table numbers and VE flow faces will remain and diffusivities will not be altered.
**Note:** When the Local Grid Refinement option is employed, the EDITNNCR keyword cannot be used within local grids.

If a non-neighbor connection to be edited cannot be found, then a warning message is issued and the EDITNNCR line ignored.

**Example**

```
EDITNNCR
 IX IY IZ JX JY JZ TRANS IST1 IST2 IPT1 IPT2 /
* ...
/
```
EHYSTR  

Hysteresis parameters and model selection

This keyword should be used if the hysteresis option has been requested (item HYSTER in keyword SATOPTS in the RUNSPEC section). It is followed by a line containing some or all of the following items, terminated with a slash (/)

If the ODD3P relative permeability and capillary pressure hysteresis model has been specified this keyword should not be used.

1. Curvature parameter for capillary pressure hysteresis.
   It must be greater than zero, and should normally lie in the range 0.05 to 0.1.
   DEFAULT: 0.1

2. Integer flag to select model for relative permeability hysteresis (capillary pressure hysteresis always uses the Killough model):
   Water wet hysteresis models:
   0 Carlson’s hysteresis model used for the non-wetting phase(s), drainage (SATNUM) curve used for the wetting phase.
   1 Carlson’s hysteresis model used for the non-wetting phase(s), imbibition (IMBNUM) curve used for the wetting phase.
   2 Killough’s hysteresis model used for the non-wetting phase(s), drainage (SATNUM) curve used for the wetting phase.
   3 Killough’s hysteresis model used for the non-wetting phase(s), imbibition (IMBNUM) curve used for the wetting phase.
   4 Killough’s hysteresis model used for both wetting and non-wetting phases.

   Oil wet to water hysteresis models:
   5 Carlson’s non-wetting model for gas and water phases (if present), drainage (SATNUM) curve used for the oil phase.
   6 Killough’s non-wetting model for the gas and water phases (if present), drainage (SATNUM) curve used for the oil phase.
   7 Killough’s hysteresis model used for non-wetting gas and water phases (if present) and the wetting oil phase.

Water wet hysteresis models:

   8 Jargon’s hysteresis model used for the non-wetting phase(s), drainage (SATNUM) curve used for the wetting phase.

   9 Jargon’s hysteresis model used for the non-wetting phase(s), imbibition (IMBNUM) curve used for the wetting phase.

   -1 Equilibration only option:

ECLIPSE Reference Manual
Option to equilibrate the model using the drainage curve (SATNUM) but to run the simulation using the imbibition curve (IMBNUM).

Hysteresis is not accounted for during the simulation; the drainage capillary pressure curves are used for equilibration but the simulation advances using the imbibition relative permeability and capillary pressure curves.

**Note:** This option is not recommended. If different capillary pressure curves are used then the model will not be in initial equilibrium.

---

3. Curvature parameter for Killough’s wetting phase relative permeability hysteresis. It must be a positive number.
   This item is ignored if item 2 is set to anything other than 4 or 7.
   DEFAULT: 1.0

4. Modification parameter for the trapped non-wetting phase saturation in the Killough model.
   If this parameter is zero (the standard Killough Model) or set to a small value, the trapped saturation is close to the maximum attained saturation when the maximum itself is small. This can lead to convergence problems.
   DEFAULT: 0.1

5. Flag limiting hysteresis to either relative permeability or capillary pressure only.
   **BOTH** Apply the hysteresis model to both the relative permeability and the capillary pressure curves, using the model selected in item 2 for the relative permeability and the Killough model for the capillary pressures.
   **PC** Apply the hysteresis model to the capillary pressure only, using the Killough capillary pressure hysteresis model.
   **KR** Apply the hysteresis model to the relative permeability only, using the model selected in item 2.
   DEFAULT: BOTH
   When the hysteresis model is not applied, the relative permeability or the capillary pressure is taken from the drainage curve (keyword SATNUM).

6. Flag controlling the shape of the capillary pressure scanning curves when secondary reversals occur.
   This flag only applies to capillary pressure hysteresis (with the Killough model).
   **RETR** When an imbibition process on a scanning imbibition curve is reversed into a drainage process, the secondary drainage process will re-traverse the same scanning curve.
   **NEW** When an imbibition process on a scanning imbibition curve is reversed into a drainage process, the secondary drainage will follow a new scanning drainage curve described by the Killough model. If further reversals are encountered, then a new scanning curve is constructed each time.
7. Flag controlling whether the initial fluid mobility correction (flag MOBILE in keyword EQLOPTS) or the transition zone calculation (keyword TZONE) applies just to the drainage curve or to both the drainage and imbibition curves.

**DRAIN** The end-points of the drainage curves only are modified to achieve the initial fluid mobility correction in each grid block.

**BOTH** The end-points of both the drainage and imbibition curves are modified to achieve the initial fluid mobility correction in each grid block.

**DEFAULT:** DRAIN

---

**ECLIPSE 100**

For the initial fluid mobility correction to be applied to the imbibition curves, the EQLOPTS option MOBILE must also be specified.

8. Flag selecting the wetting phase, between oil and gas, in three-phase cases. In two-phase gas-oil systems, oil is always wetting to gas. The oil-gas relative permeability is then calculated with the method selected in item 2.

**OIL** Oil is treated as wetting to gas in three-phase cases. The oil-gas relative permeability can then be calculated using the drainage (SATNUM) curve (option 0 and 2 and 8), or the imbibition (IMBNUM) curve (option 1 and 3 and 9), or using the Killough’s method (option 4).

**GAS** Oil is treated as non-wetting to gas in three-phase cases. The oil-gas relative permeability can then be calculated using the Carlson’s method (option 0 and 1), or the Killough’s method (option 2 and 3 and 4), or using the Jargon’s method (option 8 and 9).

In a three-phase system, oil is always default wetting phase.

---

**Note:** The choice between oil and gas as the wetting phase only applies to the modeling of the oil relative permeability to gas. The gas relative permeability is modeled as a non-wetting phase irrespective of the value ascribed here.

9. Flag selecting the BAKER1 or BAKER2 relative permeability models with hysteresis for the oil-phase case. The two phase relative permeability is then calculated with the method selected in the item 2.

**DEFAULT:** NO

10. Flag selecting the BAKER1 or BAKER2 relative permeability models with hysteresis for the gas-phase case. The two phase relative permeability is then calculated with the method selected in the item 2.

**DEFAULT:** NO

11. Flag selecting the BAKER1 or BAKER2 relative permeability models with hysteresis for the water-phase case. The two phase relative permeability is then calculated with the method selected in the item 2.

**DEFAULT:** NO

12. A threshold saturation of Killough’s hysteresis model for both wetting and non-wetting phases. This threshold allows better control of the numerical sensitivity of the system, preventing it from being too unstable.

**DEFAULT:** 0.0
13. Integer flag which can be used to specify that a modification is to be made to the wetting phase relative permeability scanning curve to mitigate the departure of the scanning curve from the region enclosed by the drainage and imbibition curves. This can sometimes occur if the initial gradients of the relative permeability curves are small. This modification is only applicable to the Killough wetting phase relative permeability hysteresis model, that is, if item 2 of this keyword set to 4 or 7, otherwise it will be ignored. See “Relative Permeability Hysteresis in the Wetting Phase” in the ECLIPSE Technical Description for more information.

DEFAULT: 0

Notes

Water wet system

- Item 8 set to gas: water is wetting to oil and wetting to gas; oil is non-wetting to water and non-wetting to gas; gas is non-wetting to water and wetting to oil. This means that in terms of wettability the phases can be ordered as W>G>O (where > means “is wetting”).
- Item 8 set to oil: water is wetting to oil and wetting to gas; oil is non-wetting to water and wetting to gas; gas is non-wetting to water and non-wetting to oil. Similarly we have W>O>G.

Oil wet system

- Item 8 set to gas: water is non-wetting to oil and non-wetting to gas; oil is wetting to water and non-wetting to gas; gas is wetting to water and wetting to oil. The wettability order is then G>O>W.
- Item 8 set to oil: water is non-wetting to oil and wetting to gas; oil is wetting to water and wetting to gas; gas is non-wetting to water and non-wetting to oil. The wettability order is O>W>G.

Note: Pre-2002A: Oil was always treated as non-wetting to gas, whatever the wettability of the system modeled.

2002A onwards: Oil can be treated also as wetting to gas with item 8, whatever the wettability of the system modeled.

The picture below depicts the segregation order of the phases with respect to their wettability as described above.

![Segregation order of phases](#)

Figure 3.4. Segregation order of phases

The hysteresis models are described in more detail in the "Hysteresis" and "Hysteresis in WAG floods" sections in the ECLIPSE Technical Description. Killough's model for capillary pressure and relative permeability hysteresis is described in his paper (1976) [Ref. 64]. See also keyword HYSTCHCK.
The default values will be used if \texttt{EHYSTR} is not entered in a run that uses the Hysteresis option. Note that for ECLIPSE 100, the default value of item 2 is 0.

For the 2014.1 version, if hysteresis is specified via the \texttt{HYSTER} option of the \texttt{SATOPTS} keyword in the \texttt{RUNSPEC} section, the default values will be used if \texttt{EHYSTR} is not entered. Prior to the 2014.1 version, if the \texttt{HYSTER} option of \texttt{SATOPTS} was specified but \texttt{EHYSTR} omitted, no hysteresis was selected. Pre-2014.1 behavior can be restored by setting item 320 of the \texttt{OPTIONS3} keyword to 1. Note that for ECLIPSE 300, the default value of item 2 is 2.

\section*{Example}

Killough’s hysteresis model applied to both capillary pressure and relative permeability, for the non-wetting phase on a water-wet system:

\begin{verbatim}
EHYSTR 0.07 2 1.0 0.1 BOTH RETR  /
\end{verbatim}

Killough’s hysteresis model used for relative permeability only. Hysteresis is implemented in both wetting and non-wetting phases for an oil-wet system, with the correction for the wetting-phase relative-permeability scanning curve:

\begin{verbatim}
EHYSTR 0.07 7 1.0 0.1 KR RETR 6* 1 /
\end{verbatim}
**EHYSTRR**

**Hysteresis parameters by region**

This keyword is entirely optional and allows the hysteresis curvature parameters to be entered on a saturation region basis. The keyword should be followed by NTSFUN records (see item 1 of keyword **TABDIMS** in the RUNSPEC section), each terminated by a slash (/).

Each record consists of the following items of data:

1. Curvature parameter for capillary pressure hysteresis.
   
   It must be greater than zero, and should normally lie in the range 0.05 to 0.1.
   
   DEFAULT: 0.1.

2. Curvature parameter for Killough’s wetting phase relative permeability hysteresis.
   
   It must be a positive number. This item will be ignored if item 2 of the **EHYSTR** keyword is set to anything other than 4.
   
   DEFAULT: 1.0.

3. Modification parameter for the trapped non-wetting phase saturation in the Killough Model.
   
   If this parameter is zero (the standard Killough model) or set to a small value, the trapped saturation will be close to the maximum attained saturation when the maximum itself is small. This can lead to convergence problems.
   
   DEFAULT: 0.1.

The three data items for each region are the same as data items 1, 3 and 4 of the **EHYSTR** keyword. If the **EHYSTRR** keyword is present this overrides the data supplied in the **EHYSTR** keyword even if the **EHYSTRR** data is defaulted.

The **EHYSTRR** data for a cell is taken from the record corresponding to the drainage region number (**SATNUM**). The data associated with the imbibition region numbers (**IMBNUM**) will be ignored.

**Example**

```
---
-- NTSFUN=4
---
EHYSTRR
0.05 1.0 0.1 /
0.07 1.0 0.1 /
0.1 1.0 0.1 /
0.1 1.0 0.1 /
```
END

Logical end of input file

This keyword terminates the input of data.

The **END** keyword need not be the actual end of the input file. However, if no **END** keyword is provided, one is generated at the end of the input data file.

**END** contains no items.

---

**Keywords**

<table>
<thead>
<tr>
<th>x ECLIPSE 100</th>
</tr>
</thead>
<tbody>
<tr>
<td>x ECLIPSE 300</td>
</tr>
<tr>
<td>SPECIAL</td>
</tr>
<tr>
<td>x RUNSPEC</td>
</tr>
<tr>
<td>x GRID</td>
</tr>
<tr>
<td>x EDIT</td>
</tr>
<tr>
<td>x PROPS</td>
</tr>
<tr>
<td>x REGIONS</td>
</tr>
<tr>
<td>x SOLUTION</td>
</tr>
<tr>
<td>x SUMMARY</td>
</tr>
<tr>
<td>x SCHEDULE</td>
</tr>
</tbody>
</table>
ENDACTIO Indicates end of ACTION keyword sequence

The ENDACTIO keyword may only be used after an ACTION, ACTIONG, ACTIONR or ACTIONW keyword has been entered. It indicates the end of the set of SCHEDULE section keywords associated with the preceding ACTIONG, ACTIONR or ACTIONW keyword.

The set of keywords between the ACTIONG, ACTIONR, ACTIONW and ENDACTIO keywords are processed when the condition defined in the ACTIONG, ACTIONR, or ACTIONW keyword has been satisfied.

The ENDACTIO keyword has no associated data.
The ENDBOX keyword has no associated data.

It causes the input box to be reset so that it encompasses the entire grid.

Note that at the end of each input section, the program effectively supplies an ENDBOX and resets the input box.

See also keywords BOX, ADD, COPY, EQUALS, MULTIPLY, MINVALUE and MAXVALUE.

Example

Thus, for an 11 by 19 by 4 grid, it has the same effect as:

```
BOX
1 11 1 19 1 4 /
```
ENDDYN  Indicates end of DYNAMICR keyword sequence

The ENDDYN keyword may only be used after a DYNAMICR keyword has been entered. It indicates the end of the set of conditions associated with the preceding DYNAMICR keyword.

The ENDDYN keyword has no associated data.
**ENDFIN**

**Terminates data for a local grid refinement**

In the **GRID** section, the **ENDFIN** keyword tells the program to finish reading **GRID** data for the local grid named in the previous **CARFIN**, **RADFIN**, **RADFIN4** or **REFINE** keyword. Any subsequent **GRID** section keywords are taken to apply to the global grid, until another local grid is specified by **CARFIN**, **RADFIN**, **RADFIN4** or **REFINE**.

It is not necessary to insert **ENDFIN** between successive **RADFIN**, **RADFIN4**, **CARFIN** or **REFINE** keywords. The primary purpose of **ENDFIN** is to revert the program to reading data for the global grid system.

In the **EDIT**, **PROPS**, **REGIONS**, **SOLUTION** and **SCHEDULE** sections, the **ENDFIN** keyword tells the program that subsequent data no longer applies to the local grid named in the previous **REFINE** keyword.

The **ENDFIN** keyword has no associated data.

*Note:* The **RADFIN4** keyword is not available in ECLIPSE 300.

Local grid refinement is described in "Local grid refinement and coarsening" in the **ECLIPSE Technical Description**.

**Example**

In the **GRID** section:

```
CARFIN -- NAME   I1-I2 J1-J2 K1-K2 NX NY NZ NWMAX--
       LGR1  2  2  3  3  4  5  5  8     5    /  
PORO  200*0.2 /  
EQUALS  
   PERMX 500 /  
   PERMY 500 /  
   PERMZ 50 /  
/  
ENDFIN
```

---

**Keywords**

- ECLIPSE 100
- ECLIPSE 300
- SPECIAL
  - RUNSPEC
  - GRID
  - EDIT
  - PROPS
  - REGIONS
  - SOLUTION
  - SUMMARY
  - SCHEDULE
**ENDINC**

**Logical end of include file**

The **ENDINC** keyword terminates the input of data from an **INCLUDE** file, returning control to the reading of the main file.

This keyword should not normally be entered, as an **ENDINC** is generated automatically at the end of the included file. It may, however, be used to end the reading of an included file before the actual end of file.
End point scaling versus depth region numbers

The keyword should be followed by one integer for every grid block in the current input box, specifying the end point scaling versus depth table region to which it belongs. The region number should not be less than 1 or greater than NTENDP (item 3 in keyword ENDScale). The data must be terminated by a slash (/).

The region number specifies which end point scaling versus depth table (input using ENPTVD and/or ENKRVD) should be used to calculate the saturation table end points for each grid block. The end point scaling option should be activated by specifying keyword ENDScale in RUNSPEC.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

See "Saturation table scaling" in the ECLIPSE Technical Description for more information.

Examples

Example 1

With NTENDP=4; NDIVIX=8, NDIVIY=6, NDIVIZ=5 specified in RUNSPEC keyword DIMENS; and no input BOX set:

```
ENDNUM
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4 /
```

Example 2

- A 12 x 4 x 5 grid
- Layers 1, 2, 3 assigned to end point scaling region number 1
- Layers 4, 5 assigned to end point scaling region number 2

```
ENDNUM
144*1 96*2 /
```
Use saturation table end-point scaling

This indicates that the saturation table end-point scaling option is to be used. It is often convenient to scale the end-points of the relative permeability curves for each cell. This option is enabled by the ENDScale keyword. Table end-points can then be entered cell by cell (keywords such as SWL, SWCR, SWU, KRW and PCW) or with respect to depth (ENPTVD, ENKRVD and ENPCVD).

The keyword should be followed by some or all of the following data items, indicating whether the end-point scaling should be directional and irreversible, and setting the dimensions of the end-point scaling tables.

The data should be terminated by a slash (/).

1. Directional end-point scaling switch

   DIRECT Indicates that the saturation table end-point scaling option is directional (that is, different saturation table end-points may be applied to saturation tables for flow in the X, Y or Z directions). End-point saturations are entered either under the appropriate directions (X,Y,Z) associated with the keywords SWL, SWCR, SWU, SGL, SSCR, SGU, SOWCR, SOGR in the PROPS section, or using the keywords ENPTVDX, ENPTVDY, ENPTVDZ in the PROPS section and ENDNUM in the REGIONS section. If the end-point relative permeabilities are also scaled, the directional forms of keywords KRW, KRG, KRO or ENKRVD must be entered.

   NODIR The saturation table end-point scaling is not directional. The same saturation table is used for flow in the X, Y or Z directions.

   DEFAULT: NODIR

2. Irreversible end-point scaling switch

   IRREVERS Indicates that the saturation table end-point scaling is irreversible (that is different saturation end points are to be used depending on whether flow is from I to I+1 or from I to I-1). Irreversible end-point saturations are entered either under the appropriate directions (+X, -X, +Y, -Y, +Z, -Z) associated with the keywords SWL, SWCR, SWU, SGL, SSCR, SGU, SOWCR, SOGR in the PROPS section, or using the keywords ENPTVDX, ENPTVDX-, ENPTVDY, ENPTVDY-, ENPTVDZ, ENPTVDZ- in the PROPS section and ENDNUM in the REGIONS section. If the end-point relative permeabilities are also scaled, the positive and negative directional forms of keywords KRW, KRG, KRO or ENKRVD must be entered.

   REVERS The end-point scaling is reversible. The same table is used whether the flow is from I to I+1 or from I to I-1.

   The IRREVERS option can only be specified if DIRECT is also specified.

   DEFAULT: REVERS

3. The number of saturation end-point versus depth tables (see keywords ENPTVD, ENKRVD in the PROPS section).

   This quantity may also be defined by item 8 of TABDIMS; if it is set in both places, ECLIPSE will use the maximum of the two values.

   DEFAULT: 1
4. The maximum number of nodes in any saturation end-point versus depth table (see keywords ENPTVD, ENKRVD in the PROPS section).

**DEFAULT:** 20

**ECLIPSE 300**

5. Options for combining the temperature-independent end-point scaling arrays (keywords such as SWL, SWCR, SWU, KRW and PCW) with the temperature-dependent end-point tables (ENPTVT, ENKRVT and ENPCVT).

0 Combination not permitted. ENSCALE and ENPTVT, ENKRVT and ENPCVT are mutually exclusive.

1 Temperature-independent grid-block end-points are superseded by temperature-dependent table end-points where non-defaulted table columns are specified.

2 Temperature-independent grid-block end-points are combined with temperature-dependent table end-points, where non-defaulted table columns are specified, using a scaling factor such that the resulting value at the lowest temperature in the table will correspond to the original grid block value.

**DEFAULT:** 0

The default option corresponds to the only option available prior to the 2011.1 version of ECLIPSE 300 for which the two end-point scaling mechanisms are mutually exclusive.

*Note:* The modeling of temperature-dependent end-points is only available for the THERMAL option. For non-thermal models item 5 will be ignored.

See "Saturation table scaling" in the ECLIPSE Technical Description for more information.

**ECLIPSE 100**

End-point scaling cannot be used in conjunction with the vertical equilibrium model (keyword VE).

**ECLIPSE 300**

The directional and reversible end-point scaling options are not available with the ODD3P relative permeability and capillary pressure hysteresis model.

**ECLIPSE 300**

Note this keyword had no items in pre-99A versions of ECLIPSE 300; existing datasets with this format will continue to run with no apparent difference.

*Note:* When directional forms of the keywords are used, the isotropic form is still required for use in equilibration and well block flows.

**Example**

End-point scaling is directional and irreversible.

```
ENDSCALE
DIRECT   IRREVERS   /
```

**Keywords**
**ENDSKIP**

**End skipping of keywords**

This keyword terminates the skipping of keywords initiated by the `SKIP`, `SKIP100` or `SKIP300` keywords. An instance of the ENDSKIP keyword must be provided for each of the SKIP, SKIP100 and SKIP300 keywords.
ENKRVC Relative permeabilities at saturation end point versus composition

The data consists of tables of relative permeabilities versus composition, one for each saturation table region, as specified in TABDIMS item 1. Each table consists of 8 columns of data (described below), and must be terminated by a slash (/).

Column

1. Molar concentration of the component identified with the EPSCOMP keyword (moles per reservoir volume). Values should increase monotonically down the column. Component molar concentrations can be output in the SUMMARY section using the BMLSC and BMWAT keywords, and in the SOLUTION and SCHEDULE section using the MLSC and MWAT mnemonics in the RPTRST, RPTSOL and RPTSCHED keywords.

   UNITS: kg-M/rm³ (METRIC), lb-M/rb (FIELD), gm-M/rcc (LAB), kg-M/rm³ (PVT-M)

2. Maximum water relative permeability.
3. Maximum gas relative permeability.
4. Maximum oil relative permeability.
5. Water relative permeability at critical oil saturation
6. Gas relative permeability at critical oil saturation
7. Oil relative permeability at an oil saturation = 1 - minimum water saturation - critical gas saturation.
8. Oil relative permeability at an oil saturation = 1 - critical water saturation - minimum gas saturation.

There must be the same number of entries in each column of a given table. The number of rows should not exceed the maximum number of entries in a quantity versus composition table, EQLDIMS item 3.

If all the permeability values in a column are defaulted then the values will be taken from the corresponding saturation function table. However, if a permeability value is defaulted whilst for other molar densities in the column values have been specified, the defaulted points will be filled via linear interpolation within the table and constant extrapolation at the ends.

Note: The ENKRVC keyword is only allowed in compositional simulations.

Note: By default, only one of the following end-point scaling methods can be used within a simulation: conventional end-point scaling initiated with the ENDScale keyword, temperature dependent end-point scaling or composition dependent end-point scaling. This behavior can be controlled using argument 5 of the ENDScale keyword.

Example

With NTDFUN=1, reducing the oil relative permeability in the presence of the nominated component.

<table>
<thead>
<tr>
<th>ENKRVC</th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>5*</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1.0</td>
<td>5*</td>
<td>0.5</td>
<td>0.5</td>
</tr>
</tbody>
</table>
/
ENKRVD  Relative permeability end point versus depth tables

The different forms of this keyword allow the user to specify the depth variation of the maximum relative permeability for the three phases. These keywords complement the ENPTVD keyword, which specifies how the end point saturation values vary with depth.

Seven keywords are provided to allow the relative permeability end points within each grid cell to be scaled independently. These keywords are:

- ENKRVD  Relative permeability end points for the non-directional saturation function tables
- ENKRVDX  Relative permeability end points for the +X face of each grid cell
- ENKRVDX- Relative permeability end points for the -X face of each grid cell
- ENKRVDY  Relative permeability end points for the +Y face of each grid cell
- ENKRVDY- Relative permeability end points for the -Y face of each grid cell
- ENKRVDZ  Relative permeability end points for the +Z face of each grid cell
- ENKRVDZ- Relative permeability end points for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the relative permeability end points for flows in the +R, -R, +T, -T, +Z, -Z directions respectively.

The keyword ENDScale in the RUNSPEC section should also be specified.

The data comprises NTENDP (item 3 of keyword ENDScale) tables of relative permeability end points vs. depth, one for each end point scaling region. Each table consists of eight columns of data, and must be terminated by a slash (/):

Column:
1. Depth values. The values should increase monotonically down the column.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
2. The corresponding values of the maximum water relative permeability.
3. The corresponding values of the maximum gas relative permeability.
4. The corresponding values of the maximum oil relative permeability.
5. Water relative permeability at the critical oil (or gas) saturation.
6. Gas relative permeability at the critical oil (or water) saturation.
7. Oil relative permeability at the critical gas saturation.
8. Oil relative permeability at the critical water saturation.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSENDP (item 4 of keyword ENDScale). If an end point is defaulted for all depth values, in a problem where a value could correctly be defined, then the end point for each cell in the same scaling region will be the value in the appropriate saturation function table. However if an end point is defaulted at a certain depth but has been defined at other depths, the end point vs. depth table will be ‘filled
in' by linear interpolation within the table and constant extrapolation at the ends of the table. End point values entered for absent phases are ignored.

**Note:** Versions of ECLIPSE 100 prior to 93A only require 4 columns of data. Hence the old datasets will require 4 extra default values on each row to run on version 93A or later. See the second example below.

If both directional and irreversible end-point scaling is requested using switches 'DIRECT' and 'IRREVERS' in **ENDSCALE**, then the relative permeability end points for flow across the +X and -X faces of a grid cell are set using the keywords `ENKRVDX`, `ENKRVDEX--`. Similarly `ENKRVDY`, `ENKRVDY--` are used for the +Y, -Y faces and the keywords `ENKRVDZ`, `ENKRVDZ--` for the +Z, -Z faces. Scaling of the well connection relative permeability tables is performed using the `ENKRVD` keyword.

When the end-point scaling is directional and reversible ('DIRECT' but not 'IRREVERS' is specified in **ENDSCALE**), the keyword `ENKRVDX` defines the relative permeability end points for the +X and -X faces of each grid cell. Keywords `ENKRVDY`, `ENKRVDZ` are used for both the +Y, -Y faces and the +Z, -Z faces of each grid cell respectively. Scaling of well connection relative permeability tables is performed using the `ENKRVD` keyword.

If **ENDSCALE** is specified without either switch 'DIRECT' or 'IRREVERS', then none of the directional forms of the `ENKRVD` keyword are allowed. The `ENKRVD` keyword then defines the new relative permeability end points for flow out of each grid cell face, and the well connection relative permeabilities.

If the hysteresis model (switch 'HYSTER' in keyword **SATOPTS**) is active then `ENKRVD` data applies to the drainage curve and the **IMKRVD** keyword is available to specify the imbibition end points as a function of depth. The `IMKRVD` keyword is used in exactly the same way as `ENKRVD`.

See also keyword **ENDNUM** in the **REGIONS** section, and the keyword **ENPTVD** in the **PROPS** section.

### Examples

#### Example 1

--- **RUNSPEC section**

```
OIL
WATER
GAS
ENDSCALE
  NTENDP  NSENDP
  2*  1      3 /
```

--- **PROPS section**

```
ENKRVD
```

<table>
<thead>
<tr>
<th>Depth</th>
<th>Krw</th>
<th>Krg</th>
<th>Kro</th>
<th>water at crit oil to water</th>
<th>gas at crit oil to gas</th>
<th>oil at crit water</th>
</tr>
</thead>
<tbody>
<tr>
<td>3000.0</td>
<td>0.50</td>
<td>1.0</td>
<td>1.0</td>
<td>0.3</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>6000.0</td>
<td>0.58</td>
<td>1.0</td>
<td>1.0</td>
<td>1*</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>9000.0</td>
<td>0.66</td>
<td>0.95</td>
<td>0.95</td>
<td>0.35</td>
<td>1*</td>
<td>1*</td>
</tr>
</tbody>
</table>

#### Example 2

Old dataset conversion:

```
ENKRVD
-- Extra columns
```
<table>
<thead>
<tr>
<th>Value</th>
<th>Column 1</th>
<th>Column 2</th>
<th>Column 3</th>
<th>Column 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>3000.0</td>
<td>0.9</td>
<td>1*</td>
<td>0.9</td>
<td>4*</td>
</tr>
<tr>
<td>4000.0</td>
<td>0.95</td>
<td>1*</td>
<td>0.95</td>
<td>4*</td>
</tr>
</tbody>
</table>
ENKRVT

Relative permeabilities at saturation end point versus temperature

This keyword can only be used with the THERMAL option.

The data consists of tables of relative permeabilities versus temperature, one for each saturation table region, as specified in TABDIMS item 1. Each table consists of 8 columns of data (described below), and must be terminated by a slash (/).

Column:

1. Temperature values.
   Should increase monotonically down the column.
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
2. Maximum water relative permeability.
3. Maximum gas relative permeability.
4. Maximum oil relative permeability.
5. Water relative permeability at critical oil saturation
6. gas relative permeability at critical oil saturation
7. Oil relative permeability at an oil saturation = 1 - minimum water saturation - critical gas saturation.
8. Oil relative permeability at an oil saturation = 1 - critical water saturation - minimum gas saturation.

There must be the same number of entries in each column of a given table. This number should not be greater than the maximum number of entries in a quantity versus temperature table (EQLDIMS item 3). Each relative permeability entry should be non-negative. Default values may be entered in columns 2 to 8 as required.

When temperature-dependent end-point scaling is used independently of grid-block end-point scaling, if an end point is defaulted for all temperature values, in a model where a value could correctly be defined, the end point for each cell in the same scaling region will be the value in the appropriate saturation function table. However, when used in combination with ENDSCALE item 5 set to 1 or 2, the end point will default to the temperature-independent grid-block scaling value for the maximum water, gas and oil relative permeabilities (columns 2, 3 and 4). However, for the water and gas relative permeabilities at critical oil saturation (columns 5 and 6) and the oil relative permeabilities at residual oil saturation (columns 7 and 8), the defaults will be supplied by scaling according to the current values of the maximum relative permeabilities.

If one or more end points in a given table column are not defaulted such that the end point has been defined for at least one temperature, the end-point versus temperature table will be filled in by linear interpolation within the table and constant extrapolation at the ends of the table. End-point values entered for absent phases are ignored.

Example

```
ENKRVT
    100.0 0.50 1.00 1.0 0.3 1* 0.8 1*
    200.0 0.66 1.00 0.95 0.35 1* 0.8 1* /
```
Capillary pressures at saturation end point versus composition

The data consists of tables of capillary pressures versus composition, one for each saturation table region, as specified in \textit{TABDIMS} item 1. Each table consists of 3 columns of data (described below), and must be terminated by a slash (/).

**Column:**

1. Molar concentration of the component identified with the \texttt{EPSCOMP} keyword (moles per reservoir volume). Values should increase monotonically down the column.
   
   Component molar concentrations can be output in the \texttt{SUMMARY} section using the \texttt{BMLSC} and \texttt{BMWAT} keywords, and in the \texttt{SOLUTION} and \texttt{SCHEDULE} section using the \texttt{MLSC} and \texttt{MWAT} mnemonics in the \texttt{RPTRST}, \texttt{RPTSOL} and \texttt{RPTSCHED} keywords.

   \textbf{UNITS:} kg-M/rm$^3$ (\texttt{METRIC}), lb-M/rb (\texttt{FIELD}), gm-M/rcc (\texttt{LAB}), kg-M/rm$^3$ (\texttt{PVT-M})

2. Maximum oil-gas capillary pressure

   \textbf{UNITS:} bars (\texttt{METRIC}), psi (\texttt{FIELD}), atm (\texttt{LAB}), atm (\texttt{PVT-M})

3. Maximum water-oil capillary pressure

   \textbf{UNITS:} bars (\texttt{METRIC}), psi (\texttt{FIELD}), atm (\texttt{LAB}), atm (\texttt{PVT-M})

There must be the same number of entries in each column of a given table. The number of rows should not exceed the maximum number of entries in a quantity versus composition table, \texttt{EQLDIMS} item 3.

If all the capillary pressure values in a column are defaulted then the values will be taken from the corresponding saturation function table. However, if a capillary pressure value is defaulted whilst for other molar densities in the column values have been specified, the defaulted points will be filled via linear interpolation within the table and constant extrapolation at the ends.

**Note:** The \texttt{ENPCVC} keyword is only allowed in compositional simulations.

**Note:** By default, only one of the following end-point scaling methods can be used within a simulation: conventional end-point scaling initiated with the \texttt{ENDSCALE} keyword, temperature dependent end-point scaling or composition dependent end-point scaling. This behavior can be controlled using item 5 of the \texttt{ENDSCALE} keyword.

**Example**

With \texttt{NTSFUN}=1, modifying the water capillary pressure in the presence of the nominated component.

```
ENPCVC
0.0  1*  45
1.0  1*  500/
```
Maximum capillary pressure versus depth tables

This keyword specifies the depth variation of the maximum capillary pressure for different end-point scaling regions within the reservoir. The keyword can only be used when the end-point scaling option is active (keyword ENDScale in the RUNSPEC section).

The data comprises NTENDP (item 3 of keyword ENDScale) tables of maximum capillary pressure end points versus depth, one for each end point scaling region. Each table consists of 3 columns of data, and must be terminated by a slash (/).

Column:

1. Depth values.
   - The values should increase monotonically down the column.
   - UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

2. The corresponding values of the maximum gas-oil capillary pressure.
   - UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

3. The corresponding values of the maximum water-oil capillary pressure.
   - UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSENDP (item 4 of keyword ENDScale). If a maximum capillary pressure is defaulted for all depth values, in a problem where a value could correctly be defined, then the end point for each cell in the same scaling region will be the value in the appropriate saturation function table. However, if a maximum capillary pressure is defaulted and the end-point has been defined at other depths, the end-point versus depth table will be filled in by linear interpolation within the table and constant extrapolation at the ends of the table. End-point values entered for absent phases are ignored.

If the hysteresis model (switch 'HYSTER' in keyword SATOPTS) is active, the ENPCVD data applies to the drainage curve and the IMPCVD keyword is available to specify the imbibition end points as a function of depth. The IMPCVD keyword is used in exactly the same way as ENPCVD.

See also keyword ENNUM in the REGIONS section, and the keywords ENPTVD, ENSPCVD, ENKRVD and IMPCVD in the PROPS section.

Example

```plaintext
-----  RUNSPEC  section
OIL
WATER
GAS
ENDSCALE
--  NTENDP  NSENDP
  2* 1  2  /
-----  PROPS  section
ENPCVD
  3000.0  1.4  23.0
  9000.0  1.2  35.0  /
```

Keywords

722
ENPCVT Capillary pressures at saturation end point versus temperature

This keyword can only be used with the THERMAL option.

The data consists of tables of capillary pressures vs. temperature, one for each saturation table region, as specified in TABDIMS item 1. Each table consists of 3 columns of data (described below), and must be terminated by a slash (/).

Column:

1. Temperature values.
   Should increase monotonically down column.
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

2. Maximum oil-gas capillary pressure
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

3. Maximum water-oil capillary pressure
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be greater than the maximum number of entries in a quantity versus temperature table (EQLDIMS item 3). Each capillary pressure entry should be non-negative. Default values may be entered in columns 2 and 3 as required.

When temperature-dependent end-point scaling is used independently of grid-block end-point scaling, if an end point is defaulted for all temperature values, in a model where a value could correctly be defined, the end point for each cell in the same scaling region will be the value in the appropriate saturation function table. However, when used in combination with ENDScale item 5 set to 1 or 2, the end point will default to the temperature-independent grid-block scaling value.

If one or more end points in a given table column are not defaulted such that the end point has been defined for at least one temperature, the end point versus temperature table will be filled in by linear interpolation within the table and constant extrapolation at the ends of the table. End-point values entered for absent phases are ignored.

Example

```
ENPCVT
100.0 10.2 0.3
200.0 14.3 5.0 /
```

Keywords

723
ENPTVC specifies the composition variation of the saturation table end-points for the saturation table regions within the reservoir. The flow of each phase across each grid block face is calculated using transformed capillary pressure and relative permeability curves obtained by linearly scaling the tabulated curves between the end-points specified using the ENPTVC keyword. ENPTVC also permits scaling of the relative permeability tables used in computing the flow of phases between grid cells and well connections and the scaling of capillary pressure functions in the equilibration algorithm.

The data comprises a number of tables of saturation end-points versus composition, one for each saturation table region, as specified in TABDIMS item 1. Each table consists of 9 columns of data (described below), and must be terminated by a slash (/).

### Column:

1. Molar concentration of the component identified with the **EPSCOMP** keyword (moles per reservoir volume). Values should increase monotonically down the column.

   Component molar concentrations can be output in the SUMMARY section using the BMLSC and BMWAT keywords, and in the SOLUTION and SCHEDULE section using the MLSC and MWAT mnemonics in the RPRTRST, RPTSOL and RPTSCHED keywords.

   **UNITS**: kg-M/rm$^3$ (**METRIC**), lb-M/rb (**FIELD**), gm-M/rcc (**LAB**), kg-M/rm$^3$ (**PVT-M**)

2. Connate water saturation.

3. Critical water saturation.


5. Connate gas saturation.


7. Maximum gas saturation.

8. Critical oil-in-water saturation


There must be the same number of entries in each column of a given table. The number of rows should not exceed the maximum number of entries in a quantity versus composition table, **EQLDIMS** item 3.

Each saturation entry should be in the range 0.0 to 1.0 inclusive. If all the saturation values in a column are defaulted then the values will be taken from the corresponding saturation function table. However, if a saturation value is defaulted whilst for other molar densities in the column values have been specified, the defaulted points will be filled via linear interpolation within the table and constant extrapolation at the ends.

**Note:** The ENPTVC keyword is only allowed in compositional simulations.

**Note:** By default, only one of the following end-point scaling methods can be used within a simulation: conventional end-point scaling initiated with the ENDSCALE keyword, temperature dependent end-point scaling or composition dependent end-point scaling. This behavior can be controlled using item 5 of the ENDSCALE keyword.
Example

With NTSFUN=1, reducing the connate and critical water saturation in the presence of the nominated component.

<table>
<thead>
<tr>
<th>ENPTVC</th>
<th>0.0</th>
<th>0.2</th>
<th>0.2</th>
<th>6*</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>6*</td>
</tr>
</tbody>
</table>
**Saturation end-point versus depth tables**

The different forms of this keyword specify the depth variation of the saturation table end-points for different end-point scaling regions within the reservoir. The flow of each phase across each grid block face is calculated using transformed capillary pressure and relative permeability curves obtained by linearly scaling the tabulated curves between the end-points specified using the ENPTVD keyword. The ENPTVD keyword also permits scaling of the relative permeability tables used in computing the flow of phases between grid cells and well connections and the scaling of capillary pressure functions in the equilibration algorithm.

Seven keywords are provided to allow the saturation end-points within each grid cell to be scaled independently. These keywords are:

- **ENPTVD**: Saturation end-points for the non-directional saturation function tables
- **ENPTVDX**: Saturation end-points for the +X face of each grid cell
- **ENPTVDX-**: Saturation end-points for the -X face of each grid cell
- **ENPTVDY**: Saturation end-points for the +Y face of each grid cell
- **ENPTVDY-**: Saturation end-points for the -Y face of each grid cell
- **ENPTVDZ**: Saturation end-points for the +Z face of each grid cell
- **ENPTVDZ-**: Saturation end-points for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the saturation end-points for flows in the +R, -R, +T, -T, +Z, -Z directions respectively.

The keyword **ENDSCALE** should be specified in the RUNSPEC section.

The data comprises NTENDP (item 3 of keyword **ENDSCALE**) tables of saturation end-points vs. depth, one for each end-point scaling region. Each table consists of 9 columns of data, and must be terminated by a slash (/).

1. Depth values. The values should increase monotonically down the column.
   - **UNITS**: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
2. The corresponding values of the connate water saturation.
3. The corresponding values of the critical water saturation.
4. The corresponding values of the maximum water saturation.
5. The corresponding values of the connate gas saturation.
6. The corresponding values of the critical gas saturation.
7. The corresponding values of the maximum gas saturation.
8. The corresponding values of the critical oil-in-water saturation.
9. The corresponding values of the critical oil-in-gas saturation.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSENDP (item 4 of keyword **ENDSCALE**). Each saturation entry should be in the range 0.0 to 1.0 inclusive. If a saturation end-point is defaulted for all depth values, in a problem where a
value could correctly be defined, then the end-point for each cell in the same scaling region will be the value in the appropriate saturation function table. However if a saturation end point is defaulted and the end-point has been defined at other depths, the end-point versus depth table will be filled in by linear interpolation within the table and constant extrapolation at the ends of the table. End point values entered for absent phases are ignored.

If both directional and irreversible end-point scaling is requested using switches ‘DIRECT’ and ‘IRREVERS’ in ENDSCALE, then saturation end-points for flow across the +X and -X faces of a grid cell are set using the keywords ENPTVDX, ENPTVDX-. Similarly ENPTVDY, ENPTVDY- are used for the +Y, -Y faces and the keywords ENPTVDZ, ENPTVDZ- for the +Z, -Z faces. Scaling of the well connection relative permeability tables and capillary pressure functions for equilibration are performed using the ENPTVD keyword.

When the end-point scaling is directional and reversible (‘DIRECT’ but not ‘IRREVERS’ is specified in ENDSCALE), the keyword ENPTVDX defines the saturation end-points for the +X and -X faces of each grid cell. Keywords ENPTVDY, ENPTVDZ are used for both the +Y, -Y faces and the +Z, -Z faces of each grid cell respectively. Scaling of well connection relative permeability tables and capillary pressure tables for equilibration are performed using the ENPTVD keyword.

If ENDSCALE is specified without either switch ‘DIRECT’ or ‘IRREVERS’, then none of the directional forms of the ENPTVD keyword are allowed. The ENPTVD keyword then defines the new saturation end-points for flow out of each grid cell face, for well connection relative permeabilities and for capillary pressure evaluations in the equilibration algorithm.

If the Hysteresis Model (switch ‘HYSTER’ in keyword SATOPTS) is active then ENPTVD data applies to the drainage curve and the IMPTVD keyword is available to specify the imbibition end-points as a function of depth. The IMPTVD keyword is used in exactly the same way as ENPTVD.

See also keyword ENNUM in the REGIONS section, and the keywords ENKRVD and IMPTVD in the PROPS section.

Example

```
----- RUNSPEC section
OIL
WATER
GAS
ENDSCALE
   -- NTENDP  NSENDP
   2*   1       2   /
----- PROPS section
ENPTVD
  3000.0  0.20  0.20  1.0  0.0  0.04  1.0  0.18  0.22
  9000.0  0.22  0.22  1.0  0.0  0.04  1.0  0.18  0.22 /
```

Keywords
ENPTVT

**Saturation end-point versus temperature**

This keyword can only be used with the THERMAL option.

ENPTVT specifies the temperature variation of the saturation table end-points for the saturation table regions within the reservoir. The flow of each phase across each grid block face is calculated using transformed capillary pressure and relative permeability curves obtained by linearly scaling the tabulated curves between the end-points specified using the ENPTVT keyword. ENPTVT also permits scaling of the relative permeability tables used in computing the flow of phases between grid cells and well connections and the scaling of capillary pressure functions in the equilibration algorithm.

The data comprises a number of tables of saturation end-points versus temperature, one for each saturation region, as specified in TABDIMS item 1. Each table consists of 9 columns of data (described below), and must be terminated by a slash (/).

1. Temperature: should increase monotonically down column.
   - UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
2. Connate water saturation.
3. Critical water saturation.
5. Connate gas saturation.
7. Maximum gas saturation.

There must be the same number of entries in each column of a given table. This number should not be greater than the maximum number of entries in a quantity versus temperature table (EQLDIMS item 3). Default values may be entered in columns 2 to 9 as required.

Each saturation entry should be in the range 0.0 to 1.0 inclusive when used to specify the temperature-dependent value directly. However, when used with ENDScale item 5 set to 2, the values can exceed 1.0.

When temperature-dependent end-point scaling is used independently of grid-block end-point scaling, if an end-point is defaulted for all temperature values, in a model where a value could correctly be defined, the end-point for each cell in the same scaling region will be the value in the appropriate saturation function table. However, when used in combination with ENDScale item 5 set to 1 or 2, the end-point will default to the temperature-independent grid-block scaling value.

If one or more end-points in a given table column are not defaulted such that the end-point has been defined for at least one temperature, the end-point versus temperature table will be filled in by linear interpolation within the table and constant extrapolation at the ends of the table. End-point values entered for absent phases are ignored.

**Example**

```
ENPTVT
100.0  0.20  0.20  1.0  0.0  0.04  1.0  0.18  0.22
200.0  0.22  0.22  1.0  0.0  0.04  1.0  0.18  0.22 /
```
**ENSPCVD Scaled saturations for P_c curves versus depth tables**

This keyword scales the connate water or gas end-point of the capillary pressure curves without scaling the corresponding oil relative permeability curves. The connate saturations are entered as functions of depth for different end-point scaling regions within the reservoir. The keyword can only be used when the end-point scaling option is active (keyword `ENDSCALE` in the RUNSPEC section).

The data comprises `NTENDP` (item 3 of keyword `ENDSCALE`) tables connate saturation end-points vs. depth, one for each end-point scaling region. Each table consists of 3 columns of data, and must be terminated by a slash `/`.

1. Depth values.
   
   The values should increase monotonically down the column.
   
   **UNITS**: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

2. The corresponding values of connate gas saturation.

3. The corresponding values of connate water saturation.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than `NSENDP` (item 4 of keyword `ENDSCALE`). If a saturation end-point is defaulted for all depth values, in a problem where a value could correctly be defined, then the end-point for each cell in the same scaling region will be the value in the appropriate saturation function table. However, if a saturation end point is defaulted and the end-point has been defined at other depths, the end-point versus depth table will be filled in by linear interpolation within the table and constant extrapolation at the ends of the table. End point values entered for absent phases are ignored.

If the hysteresis model (switch 'HYSTER' in keyword `SATOPTS`) is active, the ENSPCVD data applies to the drainage curve and the `IMSPCVD` keyword is available to specify the imbibition end-points as a function of depth. The IMSPCVD keyword is used in exactly the same way as ENSPCVD.

See also keyword `ENDNUM` in the REGIONS section, and the keywords `ENPTVD, ENPCVD, SWLPC` and `SGLPC` in the PROPS section.

**Example**

```
---- RUNSPEC section
OIL
WATER
GAS
ENDSCALE
-- NTENDP  NSENDP
  2*  1  2 /
---- PROPS section
ENSPCVD
 3000.0  1*  0.25
 9000.0  1*  0.37 /
```
Specify which equation of state is to be used

If an equation of state is being used in the compositional mode, this keyword enables one of four possibilities to be chosen. The keyword is followed by an item, the first letter of which is significant.

The possible options are:

- **PR**  Peng-Robinson
- **RK**  Redlich-Kwong
- **SRK** Soave-Redlich-Kwong
- **ZJ**  Zudkevitch-Joffe-Redlich-Kwong

Details of the equations of state are discussed in "Equations of state" in the *ECLIPSE Technical Description*.

A slight modification to the Peng-Robinson equation is available, by using the **PRCORR** keyword.

If multiple equations of state are being used, an equation of state should be chosen for each reservoir EoS region. The use of multiple EoS regions is discussed in "Equations of state regions" in the *ECLIPSE Technical Description*.

**DEFAULT:** Peng-Robinson

**Examples**

**Example 1**

```plaintext
EOS
  PR /
```

**Example 2**

With three reservoir EoS regions:

```plaintext
EOS
  P /
  S /
  S /
```
**EOSNUM**

**Equation of state region numbers**

The keyword should be followed by one integer for every grid block in the current input box specifying the equation of state region to which it belongs. The region number should not be less than 1 or greater than the number of equation of state regions, specified with the 9th item in TABDIMS.

The equation of state region number specifies which equation of state is to be used for each grid block. This is described in more detail in "Equations of state regions" in the ECLIPSE Technical Description.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2).

The default EoS index is 1.

**Example**

```
EOSNUM
  144*1  96*2  /
```
Specify surface equation of state

Specify the equation of state to be used for the production system. This defaults to the equation of state used for the field, entered using keyword `EOS`.

The possible options are:

- **PR**  Peng-Robinson
- **RK**  Redlich-Kwong
- **SRK** Souve-Redlich-Kwong
- **ZJ**  Zudkevitch-Joffe-Redlich-Kwong

Details of the equations of state are discussed in "Equations of state" in the ECLIPSE Technical Description.

If multiple equations of state are being used, an equation of state may be chosen for each surface EoS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

DEFAULT: Will follow the `EOS` keyword

**Example**

```
EOSS
PR /
```
EPSCHECK  Checks temperature-dependent scaled end-points

This keyword can be used to check the consistency of the set of temperature-dependent scaled end-points, when these are being modeled via a combination of grid-block end-points (ENDSCALE keyword item 5 set to 1 or 2) and temperature-dependent table end-points (ENPTVT, ENKRVT and ENPCVT). Under these circumstances, a consistency check will be performed at each timestep for which the EPSCHECK keyword is specified. This keyword contains no items.

Example

```plaintext
EPSCHECK
```

Keywords

- EPSCHECK
- ENPTVT
- ENKRVT
- ENPCVT
- ENDSCALE
**End-point scaling component**

The `EPSCOMP` keyword specifies the component with which to perform compositional dependent end-point scaling. The nominated component is subsequently used to interpolate the `ENKRVC`, `ENPCVC`, and `ENPTVC` tables.

The data comprises `NTSFUN` rows of data (see item 1 of the `TABDIMS` keyword) each terminated by a slash (`/`).

**Example**

With `NTSFUN=1`, and specifying end-point scaling dependence on the first hydrocarbon component.

```
EPSCOMP
  1 /
```
Controls debug for end-point scaling option

This keyword can be used to write out the scaled relative permeability curves for multiple sets of grid blocks when the end-point scaling option is active (keyword ENDSCALE in the RUNSPEC section). The curves are output as tables to the Debug file, and may optionally be output in the form of a GRAF user file which can be read directly into the GRAF program.

The keyword can also be used to switch on checking of curves in hysteresis models.

The keyword consists of the following records of data, each record terminated with a slash (/).

The data items in each record are:

**Record 1**

The first record defines control parameters for the debug:

1. Table output type
   - 0 Output in tabular form
   - 1 Output as GRAF user data

   DEFAULT: 0

2. Checks whether the drainage and imbibition curves cross in hysteresis cases. The scaled table output can be suppressed to allow large grids to be checked without creating large output files.
   - 0 No check performed. Scaled tables are output for all cells.
   - 1 Check performed. The scaled table output is switched off.
   - 2 Check performed. The scaled table output is switched off, except for cells where hysteresis problems are found.
   - 3 Check performed. Scaled tables are output for all cells.

   DEFAULT: 0

The hysteresis check can only be performed with Hysteresis enabled. See keyword SATOPTS option HYSTER. It checks the relative permeability and/or capillary pressure depending on which type of hysteresis has been selected. See item 5 of keyword EHYSTR.

End the data record with a slash (/).

**Each subsequent record**

The subsequent records, up to a maximum of 50, define the grid block data, each containing the following items of data. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

Each record should consist of 6 integers and optionally a local grid name, terminated with a slash (/). The first 6 integers define the box of cells for which the debug output will be generated.

1. IX1 First cell on X axis to be included in output
2. IX2 Last cell on X axis to be included in output
3. JY1 First cell on Y axis to be included in output
4. JY2 Last cell on Y axis to be included in output
5. KZ1 First cell on Z axis to be included in output
6. KZ2 Last cell on Z axis to be included in output

The values must satisfy:
1 ≤ IX1 ≤ IX2 ≤ NDIVIX
1 ≤ JY1 ≤ JY2 ≤ NDIVIY
1 ≤ KZ1 ≤ KZ2 ≤ NDIVIZ (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS)

7. Grid name
   ' ' Output cells in global and all local grids
   GLOBAL Output cells in global grid only
   'lgr name' Output cells in the named local grid.

   DEFAULT: ' '

This item will be ignored if local grid refinement is not used (see "Local grid refinement and coarsening" in the ECLIPSE Technical Description). The output can be quite extensive, and hence only small numbers of cells should be output from large models.

Note: Use of either of EPSDBGS or EPSDEBUG keywords for a particular run is recommended, and not both. In the event of both the keywords being used, ECLIPSE will only honor the last keyword entered.

ECLIPSE 100
For parallel runs where separate debug files are written for the individual processors, the output information for one or more grid blocks may be reported in more than one debug file if the grid blocks are used as halo cells.

ECLIPSE 300
The EPSDBGS keyword is not available with the ODD3P relative permeability and capillary pressure hysteresis model.

ECLIPSE 300
The EPSDBGS keyword can also be used in the SCHEDULE section for thermal models when temperature-dependent scaled end-points are being modeled via a combination of grid-block end-points (ENDSCALE keyword item 5 set to 1 or 2) and temperature dependent table end-points (ENPTVT, ENKRVT and ENPCVT). Under these circumstances, the debug output will be reported at each timestep for which the EPSDBGS keyword is specified.

ECLIPSE 300
In parallel, a DEBUG3 keyword, possibly empty, must be present in the RUNSPEC section for the debug output files to be created properly for each task. The run may fail otherwise. Separate debug files are used for reporting end-point scaling information for grid blocks associated with individual processors.

Note: If WAG hysteresis has been specified via the WAGHYSTR keyword and if the WAG hysteresis alternative water relative permeability saturation regions WH2NUM or WH3NUM have been specified, the alternative water relative permeabilities will be reported in the scaled tables for saturation regions for which
WAG hysteresis has been applied to the water phase. This is described under "Hysteresis in WAG floods" in the ECLIPSE Technical Description.

Example

```
EPSDBGS
  --TAB  HYST
  0     0/
  -- IX1 IX2   IY1 IY2   IZ1 IZ2   LGR
  4   4     5   5     1   3      /
  3   3     7   7     1   1     'NWEST' /
/
```

Example output

The format of the output produced by this keyword has been revised for the 2013.1 version and an example of this revised output is shown below for the water-oil saturation functions. A similar set of outputs is also produced for the gas-oil saturation functions. The output consists of a header listing the set of end-points followed by a table of relative permeabilities and capillary pressure versus saturation.

The header lists the unscaled end-points, which are determined from the input saturation table functions, together with the scaled end-points which are specified via the end-point scaling arrays (or defaulted from their corresponding table end-points if not specified). The exceptions to this are the end-points SW(KRWR), SOWL, SOW(KRORW) and SOWU which are included for completeness and which are calculated as described below:

1. **SW(KRWR)** corresponds to the scaled water saturation at which the water relative permeability at the critical oil in water saturation occurs.
   - For two-point scaling (SCALECRS keyword not specified or specified with item 1 set to No), this corresponds to the saturation $S_{w} = 1 - S_{owcr} - S_{geo}$ scaled according to the linear transformation $(S_{wcr}, S_{wmax}) \Rightarrow (SWCR, SWU)$
   - For three-point scaling (SCALECRS keyword specified with item 1 set to Yes), this corresponds to the scaled saturation $SW = 1 - SOWCR - SGL$.

2. **SOWL** corresponds to the lower limit of the scaled oil-water saturation extent and is calculated according to $SOWL = 1 - SWU - SGL$.

3. **SOW(KRORW)** corresponds to the scaled oil saturation at which the oil relative permeability to water at the critical water saturation occurs.
   - For two-point scaling (SCALECRS keyword not specified or specified with item 1 set to No), this corresponds to the saturation $S_{ow} = 1 - S_{owcr} - S_{geo}$ scaled according to the linear transformation $(S_{owcr}, S_{omax}) \Rightarrow (SOWCR, SOWU)$
   - For three-point scaling (SCALECRS keyword specified with item 1 set to Yes), this corresponds to the scaled saturation $SOW = 1 - SWCR - SGL$.

4. **SOWU** corresponds to the upper limit of the scaled oil-water saturation extent and is calculated according to $SOWU = 1 - SWL$ or $SOWU = 1 - SWL - SGL$ in ECLIPSE 100, depending upon whether item 205 of the OPTIONS keyword is set to 0 or 1 respectively, and according to $SOWU = 1 - SWL - SGL$ in ECLIPSE 300.

**Keywords**

737
For illustration purposes, these four inferred end-points are shown underlined in the example on the next page which uses two-point scaling. For the gas-oil end-point scaling output, the scaled end-points SG(KRGR), SOGL, SOG(KRORG) and SOGU are calculated in an analogous manner.

The table of relative permeabilities and capillary pressure versus saturation extends from zero to unity saturation with saturation nodes specified at 5% intervals plus the lower and upper limits of the scaled saturation extents together with the critical saturations defined in the header information.

**ECLIPSE 100**

The pre-2013.1 version of the EPSDBGS output can be selected by setting item 202 of the OPTIONS keyword to 1.

**ECLIPSE 300**

The pre-2013.1 version of the EPSDBGS output can be selected by setting item 297 of the OPTIONS3 keyword to 1.

**Note:** In ECLIPSE 100, the maximum unscaled water and gas saturations are taken from the maximum saturation entries in the respective tables. However, in ECLIPSE 300, these are calculated from the connate saturations (lowest saturation table entries) of the other phases according to

\[
S_{w_{max}} = 1 - S_{ow_{co}} - S_{g_{co}}
\]

and

\[
S_{g_{max}} = 1 - S_{og_{co}} - S_{w_{co}}
\]

This can cause differences between the simulators for the unscaled end-points in cases where the maximum saturation table entries do not concur with the values inferred from the connate saturations.

### WATER/OIL END-POINT SCALING DEBUG FOR CELL (14, 8, 8) TABLE 1

<table>
<thead>
<tr>
<th>Unscaled Swco</th>
<th>0.100000</th>
<th>Scaled SWL</th>
<th>0.200000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unscaled Swco-Pcw</td>
<td>0.100000</td>
<td>Scaled SWLPC</td>
<td>0.200000</td>
</tr>
<tr>
<td>Unscaled Sw(krwr)</td>
<td>0.700000</td>
<td>Scaled SW(KRWR)</td>
<td>0.851546</td>
</tr>
<tr>
<td>Unscaled Sw max</td>
<td>1.000000</td>
<td>Scaled SWU</td>
<td>1.000000</td>
</tr>
<tr>
<td>Unscaled krwr</td>
<td>0.500000</td>
<td>Scaled KRWR</td>
<td>0.500000</td>
</tr>
<tr>
<td>Unscaled krw max</td>
<td>1.000000</td>
<td>Scaled KRW</td>
<td>1.000000</td>
</tr>
<tr>
<td>Unscaled Sow min</td>
<td>0.000000</td>
<td>Scaled SOWL</td>
<td>0.000000</td>
</tr>
<tr>
<td>Unscaled Sowcr</td>
<td>0.300000</td>
<td>Scaled SOWCR</td>
<td>0.087762</td>
</tr>
<tr>
<td>Unscaled Sow(krorw)</td>
<td>0.090000</td>
<td>Scaled SOW(KRORW)</td>
<td>0.800000</td>
</tr>
<tr>
<td>Unscaled Sow max</td>
<td>0.900000</td>
<td>Scaled SOWU</td>
<td>0.800000</td>
</tr>
<tr>
<td>Unscaled krorw</td>
<td>1.000000</td>
<td>Scaled KRORW</td>
<td>1.000000</td>
</tr>
<tr>
<td>Unscaled kro max</td>
<td>1.000000</td>
<td>Scaled KRO</td>
<td>1.000000</td>
</tr>
<tr>
<td>Unscaled Pcw max</td>
<td>20.000000</td>
<td>Scaled PCW</td>
<td>20.000000</td>
</tr>
</tbody>
</table>

### SWAT SOIL KRW KROW PCOW

<p>| 0.000000 | 1.000000 | 0.000000 | 1.000000 | 20.000000 |
| 0.050000 | 0.950000 | 0.000000 | 1.000000 | 20.000000 |
| 0.100000 | 0.900000 | 0.000000 | 1.000000 | 20.000000 |
| 0.150000 | 0.850000 | 0.000000 | 1.000000 | 20.000000 |
| 0.200000 | 0.800000 | 0.000000 | 1.000000 | 20.000000 |
| 0.250000 | 0.750000 | 0.000000 | 0.897506 | 9.687500 |
| 0.300000 | 0.700000 | 0.000000 | 0.798244 | 5.500000 |
| 0.350000 | 0.650000 | 0.000000 | 0.702215 | 4.268750 |
| 0.400000 | 0.600000 | 0.000000 | 0.610953 | 3.500000 |
| 0.450000 | 0.550000 | 0.000000 | 0.524282 | 2.818750 |
| 0.500000 | 0.500000 | 0.000000 | 0.441460 | 2.225000 |
| 0.550000 | 0.450000 | 0.000000 | 0.363536 | 1.718750 |
| 0.554638 | 0.445362 | 0.000000 | 0.356309 | 1.675272 |
| 0.600000 | 0.400000 | 0.000000 | 0.291774 | 1.300000 |
| 0.650000 | 0.350000 | 0.000000 | 0.224688 | 0.968750 |
| 0.700000 | 0.300000 | 0.000000 | 0.162490 | 0.725000 |
| 0.750000 | 0.250000 | 0.000000 | 0.109839 | 0.543750 |
| 0.800000 | 0.200000 | 0.000000 | 0.064823 | 0.375000 |
| 0.850000 | 0.150000 | 0.000000 | 0.027963 | 0.237500 |</p>
<table>
<thead>
<tr>
<th>Value 1</th>
<th>Value 2</th>
<th>Value 3</th>
<th>Value 4</th>
<th>Value 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.851546</td>
<td>0.148454</td>
<td>0.500000</td>
<td>0.027268</td>
<td>0.234022</td>
</tr>
<tr>
<td>0.900000</td>
<td>0.100000</td>
<td>0.663522</td>
<td>0.005498</td>
<td>0.125000</td>
</tr>
<tr>
<td>0.912238</td>
<td>0.087762</td>
<td>0.704595</td>
<td>0.000000</td>
<td>0.098733</td>
</tr>
<tr>
<td>0.950000</td>
<td>0.050000</td>
<td>0.831271</td>
<td>0.000000</td>
<td>0.056250</td>
</tr>
<tr>
<td>1.000000</td>
<td>0.000000</td>
<td>1.000000</td>
<td>0.000000</td>
<td>0.000000</td>
</tr>
</tbody>
</table>
EPSDEBUG  Controls debug for end-point scaling option

This keyword can be used to write out the scaled relative permeability curves for individual grid blocks when the end-point scaling option is active (keyword ENDSCALE in the RUNSPEC section). The curves are output as tables to the Debug file, and may optionally be output in the form of a GRAF user file which can be read directly into the GRAF program.

The keyword can also be used to switch on the checking of curves in hysteresis models.

See also keyword EPSDBGS which allows scaled relative permeability curves to be written for multiple sets of grid block data.

The keyword should be followed by 7 integers and optionally a local grid name, terminated with a slash (/). The first 6 integers define the box of cells for which the debug output will be generated. This box should be restricted to a single matrix or fracture porosity. The debug data is output for all matrix and fracture porosities by default. The 7th integer controls the form of the debug output.

1. **IX1** First cell on X axis to be included in output
2. **IX2** Last cell on X axis to be included in output
3. **JY1** First cell on Y axis to be included in output
4. **JY2** Last cell on Y axis to be included in output
5. **KZ1** First cell on Z axis to be included in output
6. **KZ2** Last cell on Z axis to be included in output

The values must satisfy:

\[
1 \leq IX1 \leq IX2 \leq NDIVIX \\
1 \leq JY1 \leq JY2 \leq NDIVIY \\
1 \leq KZ1 \leq KZ2 \leq NDIVIZ\]

(Where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS)

7. Table output type
   0 Output in tabular form
   1 Output as GRAF user data

DEFAULT: 0

8. Grid name
   
   - ' ' Output cells in global and all local grids
   - GLOBAL Output cells in global grid only
   - 'lgr name' Output cells in the named local grid.

DEFAULT: '

This item will be ignored if local grid refinement is not used (see "Local grid refinement and coarsening" in the ECLIPSE Technical Description). The output can be quite extensive, and hence only small numbers of cells should be output from large models.
9. Checks whether the drainage and imbibition curves cross in hysteresis cases. The scaled table output can be suppressed to allow large grids to be checked without creating large output files.

0  No check performed. Scaled tables are output for all cells

1  Check performed. The scaled table output is switched off.

2  Check performed. The scaled table output is switched off, except for cells where hysteresis problems are found.

3  Check performed. Scaled tables are output for all cells

DEFAULT: 0

The hysteresis check can only be performed with Hysteresis enabled. See keyword SATOPTS option HYSTER. It checks whether the drainage and imbibition relative permeability and/or capillary pressure curves cross, depending on which type of hysteresis has been selected. See item 5 of keyword EHyst.

Note: It is recommended to use either the EPSDEBUG or EPSDBG3 keyword for a particular run, but not both. If both the keywords are used, ECLIPSE will only honor the last keyword entered.

ECLIPSE 100

For parallel runs where separate debug files are written for the individual processors, the output information for one or more grid blocks may be reported in more than one debug file if the grid blocks are used as halo cells.

ECLIPSE 300

The EPSDBG3 keyword is not available with the ODD3P relative permeability and capillary pressure hysteresis model.

ECLIPSE 300

The EPSDEBUG keyword can also be used in the SCHEDULE section for Thermal models when temperature-dependent scaled end-points are being modeled using a combination of grid-block end-points (ENDSCALE keyword item 5 set to 1 or 2) and temperature-dependent table end-points (ENPTVT, ENKRVT and ENPCVT). Under these circumstances, the debug output will be reported at each timestep for which the EPSDEBUG keyword is specified.

ECLIPSE 300

In parallel, a DEBUG3 keyword, possibly empty, must be present in the RUNSPEC section for the debug output files to be created properly for each task. The run may fail otherwise. Separate debug files are used for reporting end-point scaling information for grid blocks associated with individual processors.

Note: If WAG hysteresis has been specified via the WAGHYSTR keyword and if the WAG hysteresis alternative water relative permeability saturation regions WH2NUM or WH3NUM have been specified, the alternative water relative permeabilities will be reported in the scaled tables for saturation regions for which WAG hysteresis has been applied to the water phase. This is described under "Hysteresis in WAG floods" in the ECLIPSE Technical Description.

Example

<table>
<thead>
<tr>
<th>EPSDEBUG</th>
<th>IX1</th>
<th>IX2</th>
<th>IY1</th>
<th>IY2</th>
<th>IZ1</th>
<th>IZ2</th>
<th>TAB</th>
<th>LGR</th>
<th>HYST</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>4</td>
<td>4</td>
<td>5</td>
<td>5</td>
<td>1</td>
<td>3</td>
<td>0</td>
<td>1*</td>
<td>0/</td>
</tr>
</tbody>
</table>

Keywords
Example output

The format of the output produced by this keyword has been revised for the 2013.1 version and an example of this revised output is shown below for the water-oil saturation functions. A similar set of outputs is also produced for the gas-oil saturation functions. The output consists of a header listing the set of end-points followed by a table of relative permeabilities and capillary pressure versus saturation.

The header lists the unscaled end-points which are determined from the input saturation table functions, together with the scaled end-points which are specified via the end-point scaling arrays (or defaulted from their corresponding table end-points if not specified). The exceptions are the end-points SW(KRWR), SOWL, SOW(KRORW) and SOWU which are included for completeness and which are calculated as described below:

1. **SW(KRWR)** corresponds to the scaled water saturation at which the water relative permeability at the critical oil in water saturation occurs.
   
   - For two-point scaling (SCALECRS keyword not specified or specified with item 1 set to No), this corresponds to the saturation $S_w = 1 - S_{owc} - S_{gco}$ scaled according to the linear transformation $(S_{wcr}, S_{wmax}) \Rightarrow (SWCR, SWU)$
   
   - For three-point scaling (SCALECRS keyword specified with item 1 set to Yes), this corresponds to the scaled saturation $SW = 1 - SOWCR - SGL$.

2. **SOWL** corresponds to the lower limit of the scaled oil-water saturation extent and is calculated according to $SOWL = 1 - SWU - SGL$.

3. **SOW(KRORW)** corresponds to the scaled oil saturation at which the oil relative permeability to water at the critical water saturation occurs.
   
   - For two-point scaling (SCALECRS keyword not specified or specified with item 1 set to No), this corresponds to the saturation $S_{ow} = 1 - S_{owc} - S_{gco}$ scaled according to the linear transformation $(S_{owcr}, S_{omax}) \Rightarrow (SOWCR, SOWU)$
   
   - For three-point scaling (SCALECRS keyword specified with item 1 set to Yes), this corresponds to the scaled saturation $SOW = 1 - SOWCR - SGL$.

4. **SOWU** corresponds to the upper limit of the scaled oil-water saturation extent and is calculated according to $SOWU = 1 - SWL$ or $SOWU = 1 - SWL - SGL$ in ECLIPSE 100, depending upon whether item 205 of the OPTIONS keyword is set to 0 or 1 respectively, and according to $SOWU = 1 - SWL - SGL$ in ECLIPSE 300.

For illustration purposes, these four end-points are shown underlined in the example over the page which uses two-point scaling. For the gas-oil end-point scaling output, the scaled end-points SG(KRGR), SOGL, SOG(KRORG) and SOGU are calculated in an analogous manner.

The table of relative permeabilities and capillary pressure versus saturation extends from zero to unity saturation with saturation nodes specified at 5% intervals plus the lower and upper limits of the scaled saturation extents together with the critical saturations defined in the header information.

**ECLIPSE 100**

The pre-2013.1 version of the EPSDEBUG output can be selected by setting item 202 of the OPTIONS keyword to 1.

**ECLIPSE 300**

The pre-2013.1 version of the EPSDEBUG output can be selected by setting item 297 of the OPTIONS3 keyword to 1.
**Note:** In ECLIPSE 100, the maximum unscaled water and gas saturations are taken from the maximum saturation entries in the respective tables. However, in ECLIPSE 300, these are calculated from the connate saturations (lowest saturation table entries) of the other phases according to:

\[ S_{w\text{max}} = 1 - S_{w\text{co}} - S_{g\text{co}} \]
\[ S_{g\text{max}} = 1 - S_{og\text{co}} - S_{w\text{co}} \]

This can cause differences between the simulators for the unscaled end-points in cases where the maximum saturation table entries do not concur with the values inferred from the connate saturations.

---

**WATER/OIL END-POINT SCALING DEBUG FOR CELL (14, 8, 8) TABLE 1**

<table>
<thead>
<tr>
<th>Unscaled</th>
<th>Unscaled</th>
<th>Unscaled</th>
<th>Unscaled</th>
<th>Unscaled</th>
<th>Unscaled</th>
</tr>
</thead>
<tbody>
<tr>
<td>Swco</td>
<td>0.100000</td>
<td>0.100000</td>
<td>0.100000</td>
<td>0.100000</td>
<td>0.100000</td>
</tr>
<tr>
<td>Swco-Cw</td>
<td>0.100000</td>
<td>0.100000</td>
<td>0.100000</td>
<td>0.100000</td>
<td>0.100000</td>
</tr>
<tr>
<td>Swcr</td>
<td>0.070000</td>
<td>0.070000</td>
<td>0.070000</td>
<td>0.070000</td>
<td>0.070000</td>
</tr>
<tr>
<td>Sw(kw)</td>
<td>1.000000</td>
<td>1.000000</td>
<td>1.000000</td>
<td>1.000000</td>
<td>1.000000</td>
</tr>
<tr>
<td>Sw max</td>
<td>0.500000</td>
<td>0.500000</td>
<td>0.500000</td>
<td>0.500000</td>
<td>0.500000</td>
</tr>
<tr>
<td>kwr</td>
<td>0.100000</td>
<td>0.100000</td>
<td>0.100000</td>
<td>0.100000</td>
<td>0.100000</td>
</tr>
<tr>
<td>kwr max</td>
<td>1.000000</td>
<td>1.000000</td>
<td>1.000000</td>
<td>1.000000</td>
<td>1.000000</td>
</tr>
<tr>
<td>Sow min</td>
<td>0.000000</td>
<td>0.000000</td>
<td>0.000000</td>
<td>0.000000</td>
<td>0.000000</td>
</tr>
<tr>
<td>Sow cr</td>
<td>0.300000</td>
<td>0.300000</td>
<td>0.300000</td>
<td>0.300000</td>
<td>0.300000</td>
</tr>
<tr>
<td>Sow(kw)</td>
<td>0.900000</td>
<td>0.900000</td>
<td>0.900000</td>
<td>0.900000</td>
<td>0.900000</td>
</tr>
<tr>
<td>Sow max</td>
<td>0.900000</td>
<td>0.900000</td>
<td>0.900000</td>
<td>0.900000</td>
<td>0.900000</td>
</tr>
<tr>
<td>krow</td>
<td>1.000000</td>
<td>1.000000</td>
<td>1.000000</td>
<td>1.000000</td>
<td>1.000000</td>
</tr>
<tr>
<td>kro max</td>
<td>1.000000</td>
<td>1.000000</td>
<td>1.000000</td>
<td>1.000000</td>
<td>1.000000</td>
</tr>
<tr>
<td>Pcw max</td>
<td>20.000000</td>
<td>20.000000</td>
<td>20.000000</td>
<td>20.000000</td>
<td>20.000000</td>
</tr>
</tbody>
</table>

---

### WATER/OIL saturations

<table>
<thead>
<tr>
<th>SWAT</th>
<th>SOIL</th>
<th>KRW</th>
<th>KROW</th>
<th>PCOW</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000000</td>
<td>1.000000</td>
<td>0.000000</td>
<td>1.000000</td>
<td>20.000000</td>
</tr>
<tr>
<td>0.050000</td>
<td>0.950000</td>
<td>0.000000</td>
<td>1.000000</td>
<td>20.000000</td>
</tr>
<tr>
<td>0.100000</td>
<td>0.900000</td>
<td>0.000000</td>
<td>1.000000</td>
<td>20.000000</td>
</tr>
<tr>
<td>0.150000</td>
<td>0.850000</td>
<td>0.000000</td>
<td>1.000000</td>
<td>20.000000</td>
</tr>
<tr>
<td>0.200000</td>
<td>0.800000</td>
<td>0.000000</td>
<td>1.000000</td>
<td>20.000000</td>
</tr>
<tr>
<td>0.250000</td>
<td>0.750000</td>
<td>0.000000</td>
<td>0.897506</td>
<td>9.687500</td>
</tr>
<tr>
<td>0.300000</td>
<td>0.700000</td>
<td>0.000000</td>
<td>0.798244</td>
<td>5.500000</td>
</tr>
<tr>
<td>0.350000</td>
<td>0.650000</td>
<td>0.000000</td>
<td>0.702215</td>
<td>4.268750</td>
</tr>
<tr>
<td>0.400000</td>
<td>0.600000</td>
<td>0.000000</td>
<td>0.610953</td>
<td>3.500000</td>
</tr>
<tr>
<td>0.450000</td>
<td>0.550000</td>
<td>0.000000</td>
<td>0.524282</td>
<td>2.818750</td>
</tr>
<tr>
<td>0.500000</td>
<td>0.500000</td>
<td>0.000000</td>
<td>0.441460</td>
<td>2.250000</td>
</tr>
<tr>
<td>0.550000</td>
<td>0.450000</td>
<td>0.000000</td>
<td>0.363536</td>
<td>1.718750</td>
</tr>
<tr>
<td>0.554638</td>
<td>0.445362</td>
<td>0.000000</td>
<td>0.356309</td>
<td>1.675272</td>
</tr>
<tr>
<td>0.600000</td>
<td>0.400000</td>
<td>0.000000</td>
<td>0.291774</td>
<td>1.300000</td>
</tr>
<tr>
<td>0.650000</td>
<td>0.350000</td>
<td>0.000000</td>
<td>0.224688</td>
<td>0.968750</td>
</tr>
<tr>
<td>0.700000</td>
<td>0.300000</td>
<td>0.000000</td>
<td>0.162490</td>
<td>0.725000</td>
</tr>
<tr>
<td>0.750000</td>
<td>0.250000</td>
<td>0.000000</td>
<td>0.109839</td>
<td>0.543750</td>
</tr>
<tr>
<td>0.800000</td>
<td>0.200000</td>
<td>0.000000</td>
<td>0.056250</td>
<td></td>
</tr>
<tr>
<td>0.850000</td>
<td>0.150000</td>
<td>0.000000</td>
<td>0.027963</td>
<td>0.375000</td>
</tr>
<tr>
<td>0.851546</td>
<td>0.148454</td>
<td>0.000000</td>
<td>0.027268</td>
<td>0.234022</td>
</tr>
<tr>
<td>0.900000</td>
<td>0.100000</td>
<td>0.000000</td>
<td>0.005498</td>
<td>0.125000</td>
</tr>
<tr>
<td>0.912238</td>
<td>0.087762</td>
<td>0.000000</td>
<td>0.005498</td>
<td>0.125000</td>
</tr>
<tr>
<td>0.950000</td>
<td>0.050000</td>
<td>0.000000</td>
<td>0.005498</td>
<td>0.125000</td>
</tr>
<tr>
<td>1.000000</td>
<td>0.000000</td>
<td>0.000000</td>
<td>0.000000</td>
<td>0.000000</td>
</tr>
</tbody>
</table>

---

**Keywords**

743
End-point and process scaling control for ODD3P option

The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) tables of end-point scaling control data, each terminated by a slash (/). Only one row per table should be entered. This keyword is only used with the ODD3P relative permeability and capillary pressure hysteresis option and is optional.

Each table consists of a single row containing 26 columns of data:

1. Oil saturation end-point Land process scaling control.
   - 0  Deselect oil saturation Land process end-point scaling.
   - 1  Select oil saturation Land process end-point scaling.
   
   DEFAULT: 0
   
   For miscible runs, a value of 1 is recommended.

2. Gas saturation end-point Land process scaling control.
   - 0  Deselect gas saturation Land process end-point scaling.
   - 1  Select gas saturation Land process end-point scaling.
   
   DEFAULT: 0
   
   For miscible runs, a value of 1 is recommended.

3. Water saturation end-point Land process scaling control.
   - 0  Deselect water saturation Land process end-point scaling.
   - 1  Select water saturation Land process end-point scaling.
   
   DEFAULT: 0
   
   For miscible runs, a value of 1 is recommended.

4. Capillary pressure scaling control.
   - 0  Deselect capillary pressure scaling.
     
     If a value of 0 is specified, no capillary pressure scaling is performed.
   - 1  Select capillary pressure scaling using reference surface tensions.
     
     If a value of 1 is specified, the capillary pressure scaling is based upon the ratios of the calculated values of oil-gas, oil-water and gas-water surface tensions to the corresponding reference surface tension values defined in columns 6, 7 and 8.
   - 2  Select capillary pressure scaling using reference surface tensions plus exponents.
     
     If a value of 2 is specified, the capillary pressure scaling is based upon the ratios of the calculated values of oil-gas, oil-water and gas-water surface tensions to corresponding reference surface tensions plus exponents.
tension values defined in columns 6, 7 and 8, raised to the corresponding exponents defined in columns 9, 10 and 11.

DEFAULT: 0

5. End-point saturation and relative permeability scaling control.

0 Deselect end-point saturation relative permeability scaling.

If a value of 0 is specified, no end-point scaling is performed.

1 Select end-point saturation and relative permeability scaling using threshold surface tensions.

If a value of 1 is specified, the end-point saturation and relative permeability scaling is based upon the calculated values of oil-gas, oil-water and gas-water surface tensions and the corresponding threshold surface tension values defined in columns 15, 16 and 17 below.

2 Select end-point saturation and relative permeability scaling using threshold surface tensions plus exponents

If a value of 2 is specified, the end-point saturation and relative permeability scaling is based upon the calculated values of oil-gas, oil-water and gas-water surface tensions, the corresponding reference surface tension values defined in columns 15, 16 and 17 below raised to the corresponding exponents defined in columns 24, 25 and 26.

3 Select end-point saturation and relative permeability scaling using threshold capillary numbers.

If a value of 3 is specified, the end-point saturation and relative permeability scaling is based upon the calculated values of oil-gas, oil-water and gas-water surface tensions, the calculated capillary numbers and the threshold capillary numbers specified in columns 15 to 20.

4 Select end-point saturation and relative permeability scaling using threshold capillary numbers plus exponents.

If a value of 4 is specified, the end-point saturation and relative permeability scaling is based upon the calculated values of oil-gas, oil-water and gas-water surface tensions, the calculated capillary numbers and the threshold capillary numbers specified in columns 15 to 20 raised to the exponents defined in columns 24, 25 and 26.

DEFAULT: 0

Relative permeability scaling is only applied for a given phase pair when the calculated surface tension is less than the threshold surface tension (options 1 and 2) or the calculated capillary number is greater than the threshold capillary number (options 3 and 4); that is, the relative permeability scaling factors are constrained to lie between 0.0 and 1.0. A small (or zero) threshold surface tension value or a large capillary number threshold value for a given phase pair, therefore, will have the effect of disabling scaling for that phase pair.


DEFAULT: 1.0

UNITS: dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT-M)

This value is only used if the capillary pressure scaling control parameter (column 4) is defined to be 1 or 2.

7. Oil-water reference surface tension.

DEFAULT: 1.0

UNITS: dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT-M)

This value is only used if the capillary pressure scaling control parameter (column 4) is defined to be 1 or 2.


DEFAULT: 1.0

This value is only used if the capillary pressure scaling control parameter (column 4) is defined to be 2.

10. Oil-water capillary pressure surface tension scaling factor exponent.

DEFAULT: 1.0

This value is only used if the capillary pressure scaling control parameter (column 4) is defined to be 2.


DEFAULT: 1.0

This value is only used if the capillary pressure scaling control parameter (column 4) is defined to be 2.

12. First constant $A_1$ in water-hydrocarbon interfacial tension correlation equation.

DEFAULT: 1.0

The hydrocarbon-water interfacial tension is calculated according to the equation:

$$\sigma_{hw} = A_1 + (\Delta \rho (A_2 + \Delta \rho A_3))$$

with $\Delta \rho$ given by:

$$\Delta \rho = \rho_w - \rho_h$$

where $\rho_w$ and $\rho_h$ are the densities of water and hydrocarbon phases (oil or gas) respectively in gm/cc.

This value is only used if the capillary pressure scaling control parameter (column 4) is defined to be greater than 0 or the end-point saturation and relative permeability scaling control parameter (column 5) is defined to be greater than 0.

13. Second constant $A_2$ in water-hydrocarbon interfacial tension correlation equation.

DEFAULT: 0.0

The hydrocarbon-water interfacial tension is calculated according to the equation:

$$\sigma_{hw} = A_1 + (\Delta \rho (A_2 + \Delta \rho A_3))$$

with $\Delta \rho$ given by:
\[ \Delta \rho = \rho_w - \rho_h \] where \( \rho_w \) and \( \rho_h \) are the densities of water and hydrocarbon phases (oil or gas) respectively in gm/cc.

This value is only used if the capillary pressure scaling control parameter (column 4) is defined to greater than 0 or the end-point saturation and relative permeability scaling control parameter (column 5) is defined to be greater than 0.

14. Third constant \( A_3 \) in water-hydrocarbon interfacial tension correlation equation.

DEFAULT: 0.0

The hydrocarbon-water interfacial tension is calculated according to the equation:

\[ \sigma_{hw} = A_1 + (\Delta \rho (A_2 + \Delta \rho A_3)) \]

with \( \Delta \rho \) given by:

\[ \Delta \rho = \rho_w - \rho_h \] where \( \rho_w \) and \( \rho_h \) are the densities of water and hydrocarbon phases (oil or gas) respectively in gm/cc.

This value is only used if the capillary pressure scaling control parameter (column 4) is defined to be greater than 0 or the end-point saturation and relative permeability scaling control parameter (column 5) is defined to be greater than 0.

15. Threshold interfacial tension or capillary number value for activating scaling on the oil phase property in an oil-gas system.

DEFAULT: 0.0

UNITS (for interfacial tension): dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT-M)

This value is only used if the end-point saturation and relative permeability scaling control parameter (column 5) is defined to be greater than 0.

16. Threshold interfacial tension or capillary number value for activating scaling on the oil phase property in an oil-water system.

DEFAULT: 0.0

UNITS (for interfacial tension): dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT-M)

This value is only used if the end-point saturation and relative permeability scaling control parameter (column 5) is defined to be greater than 0.

17. Threshold interfacial tension or capillary number value for activating scaling on the gas phase property in a gas-water system.

DEFAULT: 0.0

UNITS (for interfacial tension): dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT-M)

This value is only used if the end-point saturation and relative permeability scaling control parameter (column 5) is defined to be greater than 0.

18. Threshold capillary number value for activating scaling on the gas phase property in a gas-oil system.

DEFAULT: 0.0
This value is only used if the end-point saturation and relative permeability scaling control parameter (column 5) is defined to 3 or 4.

19. Threshold capillary number value for activating scaling on water phase property in a water-oil system.
   DEFAULT: 0.0
   This value is only used if the end-point saturation and relative permeability scaling control parameter (column 5) is defined to be 3 or 4.

20. Threshold capillary number value for activating scaling on the water phase property in a water-gas system.
   DEFAULT: 0.0
   This value is only used if the end-point saturation and relative permeability scaling control parameter (column 5) is defined to be 3 or 4.

21. Primary to hysteresis grid-block end-point rate of change.
   UNITS: 1/day (METRIC), 1/day (FIELD), 1/hr (LAB), 1/day (PVT-M)
   DEFAULT: 0.01

22. Minimum change in saturation required to signal change in saturation direction.
   DEFAULT: 0.000001 (IMPES), 0.001 (AIM, FULLIMP and IMPSAT)

23. First contact miscible control
   0  Do not check for phase label switching
   1  Check for phase label switching

   If a value of 1 is specified a check for phase label switching is performed when a first contact miscible condition occurs. In this case, in order to ensure consistency, the gas-water residual saturations will be taken from the oil-water values.

   DEFAULT: 0

24. Gas-oil end-point saturation and relative permeability surface tension and capillary number scaling factor exponent.
   DEFAULT: 1.0
   This value is only used if the end-point saturation and relative permeability scaling control parameter (column 5) is defined to be 2 or 4. For capillary number scaling this exponent is applied to both the gas-oil and oil-gas scaling factors.

25. Oil-water end-point saturation and relative permeability surface tension and capillary number scaling factor exponent.
   DEFAULT: 1.0
   This value is only used if the end-point saturation and relative permeability scaling control parameter (column 5) is defined to be 2 or 4. For capillary number scaling this exponent is applied to both the oil-water and water-oil scaling factors.

26. Gas-water end-point saturation and relative permeability surface tension and capillary number scaling factor exponent.
   DEFAULT: 1.0
This value is only used if the end-point saturation and relative permeability scaling control parameter (column 5) is defined to be 2 or 4. For capillary number scaling this exponent is applied to both the gas-water and water-gas scaling factors.

Default values (represented by 1*) may be inserted as required in columns 1 to 26. Each row may span more than one line of input data if required. When the table is read in, these defaults are replaced by the default values specified above.

The second and subsequent tables may be defaulted entirely. Defaulted tables are replaced with a copy of the previous table.

Only the tables associated with the primary saturation region numbers defined using the PSTNUM keyword are used. Tables associated with the secondary and tertiary hysteresis saturation region numbers defined using the ISTNUM and DSTNUM keywords respectively are ignored and may be defaulted.

See "Saturation functions" in the ECLIPSE Technical Description for a description of the method used to compute the oil relative permeability in blocks where all three phases are present.

The reference and exponent surface tensions and the hydrocarbon-water interfacial tension correlation coefficients defined in columns 6 to 17 above are used only when values of 1 or 2 have been specified in columns 4 or 5. These values are used both in the scaling of the values of relative permeability and capillary pressure obtained from table look-up and also in the scaling of the primary and hysteresis grid-block end-point saturations. For these options no distinction is made between oil-gas/gas-oil, oil-water/water-oil and gas-water/water-gas in applying the end-point scaling; that, is the same scaling is applied to each member of these pairs of end-points. If capillary number end-point scaling is selected, however, the six end-points are determined individually and the values in column 15 to 17 and 18 to 20 should be used to define the six threshold capillary number values.

See also the keywords PCODD3P, PCODD3PG and PCODD3PW.

Example

With NTsfUn=3 and assuming that the primary saturation region is associated with table 1:

```
EPSODD3P
--- Process Scale   Pc   Pc SurfTen Kr          1st Kr
--- Control Cntrl Ref-STen Exp-STen HW-Coef Th Sat Sat Con St
--- Lo Lq Lw Pc Kr GO OW GW GO OW GW A1 A2 A3 St dSdT dSmn Mis Ex
  1  1  1  1  0 .07 27 23 1. 1. 1.  1. 1. 1.  6* 0.01 0.001 0 3*/
```

Keywords
**Dimensions of equilibration tables**

The data consists of some or all of the following items which specify the dimensions of the equilibration tables. The data must be terminated by a slash (/).

1. **NTEQUL**, the number of equilibration regions entered using **EQLNUM** in the **REGIONS** section. Different equilibration regions may be used to initialize different parts of the reservoir that are not in mutual hydrostatic equilibrium. (See keywords **EQLNUM** in the **REGIONS** section and **EQUIL** in the **SOLUTION** section.)
   
   DEFAULT: 1

2. The number of depth nodes in any table of pressure versus depth constructed internally by the equilibration algorithm.

   (ECLIPSE 100) DEFAULT: 100
   
   (ECLIPSE 300) DEFAULT: 50

3. **NDRXVD**, the maximum number of depth nodes in any **RSVD**, **RVVD**, **RSWVD**, **RTEMPVD**, **PBVD** or **PDVD** table entered in the **SOLUTION** section to define the initial **R_s**, **R_v**, **T_r**, **P_b** or **P_d** versus depth.

   **ECLIPSE 100**
   
   **ECLIPSE 300**

   For ECLIPSE 300 this parameter also applies to:

   - The number of entries in the **ZMFVD** keyword.
   - The maximum number of rows for the end-point versus temperature keywords **ENKRVT**, **ENPCVT**, **ENPTVT**.
   - The maximum number of rows for the end-point versus composition keywords **ENKRVC**, **ENPCVC**, **ENPTVC**.

   **ECLIPSE 100** DEFAULT: 20
   
   **ECLIPSE 300** DEFAULT: 50

4. **NTTRVD**, the number of tables of initial tracer concentration versus depth (keyword **TVDP** in the **SOLUTION** section).

   DEFAULT: 1

5. **NSTRVD**, the maximum number of depth nodes in any table of initial tracer concentration versus depth (keyword **TVDP** in the **SOLUTION** section).

   **ECLIPSE 100**
   
   **ECLIPSE 300**

   DEFAULT: 20
   
   DEFAULT: 50

See "Initializing the study" in the **ECLIPSE Technical Description** for more information.

**Example**

Two equilibration regions (**EQLNUM**) with the default number of nodes in the internal pressure versus depth tables and 30 nodes in an **RSVD** table.

```
EQLDIMS
2 1* 30 /
```

**Keywords**

750
EQLDKVCR  Equilibrium deviation reaction k-values

The keyword is used to specify k-values for modeling reactions that deviate from an equilibrium state. See keyword EQLDREAC which specifies the equilibrium deviation terms. The number of k-values, N\(_k\), is specified by the third item in the REACTION keyword. K-values are specified in a similar manner to the KVCR keyword where up to five constants can be used to define each equilibrium value:

\[
K(P, T) = \left( A + B \frac{1}{P} + C \cdot P \right) \cdot e^{-D(T-E)}
\]

where

- \(T\) is the temperature (K (METRIC), °R (FIELD), K (LAB), K (PVT-M))
- \(P\) is the pressure (barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)).

The temperature and pressure values used to calculate these equilibrium values can be limited using the REACLIMS keyword.

DEFAULT: 0.0

This keyword is followed by up to five rows, each containing N\(_k\) columns of data described below. The set of records is ended by a blank record, containing only a slash.

1. A, N\(_k\) values
   
   UNITS: dimensionless

2. B, N\(_k\) values
   
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

3. C, N\(_k\) values
   
   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

4. D, N\(_k\) values
   
   UNITS: K (METRIC), °R (FIELD), K (LAB), K (PVT-M)

5. E, N\(_k\) values
   
   UNITS: K (METRIC), °R (FIELD), K (LAB), K (PVT-M)

See "Chemical reactions" in the ECLIPSE Technical Description.

If multiple equations of state are being used, only one set of data is entered for this keyword, unlike keyword KVCR, where one set of data is used for each equation of state.

**Example**

With item 3 of REACTION set to 2:

\[
\text{EQLDKVCR} \quad -- \text{K1} \quad \text{K2} \\
1.23E6 \quad 212 \\
833.4E6 \quad 155.4E3 \\
0 \quad 0 \\
16000 \quad 4000
\]
Equilibrium deviation terms for reactions

Equilibrium deviation terms can be used to model deviation from an equilibrium state. A reaction rate is multiplied by a term:

\[ |d|_n, \quad d > 0 \]
\[ -|d|_n, \quad d < 0 \]

where \( n \) is the order of the equilibrium deviation term and \( d \) is given by one of the following models:

\[ d = \theta \cdot b_o \cdot S_o \cdot (x_i y_j / K_k), \quad m = 1 \]
\[ d = \theta \cdot b_g \cdot S_g \cdot (K_k \cdot x_i y_j), \quad m = 2 \]
\[ d = \theta \cdot b_w \cdot S_w \cdot (a_i - 1 / K_k), \quad m = 3 \]
\[ d = \theta \cdot (F_k(a_i) - C_a), \quad m = 4 \]

Here:

\( \theta \) is the porosity,
\( b_o, b_g \) and \( b_w \) are the molar densities of the oil, gas and water phases,
\( S_o, S_g \) and \( S_w \) are the oil, gas and water phase saturations,
\( x_i, y_j, a_i \) and \( a_j \) are oil, gas and water component mole fractions,
\( C_a \) is the adsorbed solid concentration
\( K_k \) and \( F_k \) are equilibrium values.

The keyword is followed by a number of records, each specifying an equilibrium deviation term. The number of records is specified by the second item in the REACTION keyword. Each record contains a number of items of data and is terminated by a slash.

1. The reaction number.
2. The model number \( m \) for the deviation term, between 1 and 4, see equations 3.65 - 3.68. Models 3 and 4 can only be used with the Multiple Water Component Model, which is activated with the COMPW keyword. Model 4 can only be used with SOLID.
3. The component number \( i \).
4. The component number \( j \).
5. The equilibrium value number \( k \), which specifies either
   - a k-value \( K_k \) defined by keyword EQLDKVCR (\( m \leq 3 \)) or
   - an equilibrium function \( F_k \) defined by keyword EQLDTAB (\( m = 4 \))
6. The direction in which the reaction can proceed:
   - **FORWARD**: Reaction rate set to zero if less than zero

**Keywords**

EQLDREAC
BACKWARD  Reaction rate set to zero if more than zero

BOTH      Reaction can proceed in both directions depending on the sign of the rate.

DEFAULT: BOTH

7. The order \( n > 0 \).

DEFAULT: 1.0

The reaction can be limited to proceed in only one direction via the 6th item. However, since the derivative of the rate will generally not be smooth around zero this can cause stability issues.

The reaction stoichiometry (STOREAC and STOPROD) should be set so that component \( i \) reacts to form component \( j \) for positive reaction rates.

For \( m \leq 2 \), the component numbers \( i \) and \( j \) must be between 0 and the number of hydrocarbon components, \( 0 \leq i, j \leq N_c \). If a value of 0 is given then the mole fraction \( (x_i \text{ or } y_j) \) is replaced by a 1 in equation 3.65 and equation 3.66.

For \( m = 3 \) or \( m = 4 \) the component number \( i \) must represent a water component, \( N_c < i \leq N_c + N_w \). The component number \( j \) is not used for these models and should be set to 0.

See "Chemical reactions" in the ECLIPSE Technical Description.

Notes

- The first two models \( m \leq 2 \) are provided for thermal simulations where they can be used with non-volatile or non-condensable components. The second two models \( m \geq 3 \) can be used with water components. The keyword can be used outside these limitations, but it may compete with the flash and cause convergence problems when used with components that are not inert.

- To ensure that the reaction rate of zero-order reactants tends to zero as the reactant disappears, the reaction rate is multiplied by a factor \( m / (m + s) \) where \( m \) is the molar density of the reactant and \( s \) is given by item 4 of keyword REACOPS.

- If item 305 of keyword OPTIONS3 is set to 1, the simulator reverts to the pre-2013.1 treatment of zero-order reactants when calculating the equilibrium deviation terms.

Example

In this example the equilibrium deviation reaction rates for reactions 1 and 2 are:

\[
d = \theta \cdot b_o \cdot S_o \cdot (x_1 - y_2 / K_1) 
\]

and

\[
d = \theta \cdot b_o \cdot S_o \cdot (x_3 - 1 / K_2). 
\]

This second term can be used to model a trapped gas phase where \( y_j = 1 \) since the trapped gas phase is separate from the free gas phase. In this example, item 2 of REACTION is set to 2:

```
EQLDREAC
   -- R M I J K  DIR
   1 1 3 2 1 BOTH /
   2 1 3 0 2 BOTH /
```
EQLDTAB

Equilibrium values for water-solid deviation reactions

The keyword is used to specify tables of equilibrium values for modeling reactions that deviate from an equilibrium state. The equilibrium states are defined by the amount of adsorbed solid as a function of a water component mole fraction. The number of tables, $N_T$, is specified by the item 5 in the REACTION keyword. Each table consists of two columns of data and is terminated by a slash (/).

1. A water component mole fraction.

   The values should be between 0 and 1 and increase monotonically down the column.

2. The corresponding adsorbed solid concentration, defined as mass of solid per mass of rock.

   UNITS: kg/kg (METRIC), lb/lb (FIELD), gm/gm (LAB), kg/kg (PVT-M)

The maximum number of rows in each table is $N_{PVT}$ (item 4 of keyword TABDIMS).

The EQLDREAC keyword is required to specify the equilibrium deviation terms and the ROCKDEN keyword is required to specify the rock density.

See "Chemical reactions", "Water components" and "Solid phase" in the ECLIPSE Technical Description.

Example

```
EQLDTAB
   -- AMF EQ
   0.000   0.00000
   0.001   0.0005
   1.000   0.0005
/
```
Equilibration region numbers

The keyword should be followed by one integer for every grid block in the current input box specifying the equilibration region to which it belongs. The region number should not be less than 1 or greater than NTEQUL (item 1 in keyword EQLDIMS). All blocks with the same equilibration region number must also have the same PVT region number (see keyword PVNUM). The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

See also the keywords EQUIL, RSVD, RVVD and PBVD in the SOLUTION section.

Example

With NTEQUL=6:

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX       4   11   3   8   1   5   /
EQLNUM
8*1 8*2  8*3  8*4  8*5  8*6
8*1 8*2  8*3  8*4  8*5  8*6
8*1 8*2  8*3  8*4  8*5  8*6
8*1 8*2  8*3  8*4  8*5  8*6
8*1 8*2  8*3  8*4  8*5  8*6 /
```
**EQLOPTS**

## Options for equilibration

The keyword should be followed by up to four items, which set options for the initial equilibration algorithm. The data should be terminated by a slash (/). The possible values of the options are:

- **MOBILE** Activates the initial mobile fluid critical saturation end point correction. A fine scale Equilibration option should be used (see keyword **EQUIL**). The option should not be used in VE runs (keyword **VE**). See “Saturation table scaling” in the *ECLIPSE Technical Description* for more information.

  This option applies to **ECLIPSE 100 only**.

- **QUIESC** Enables pressure modifications to achieve initial quiescence (that is to produce a true steady state solution). A block-center equilibration (item 9 in keyword **EQUIL** set to 0) is always quiescent, but the more accurate options are not absolutely quiescent. If this switch is set, modifications are applied to the phase pressures to make the initial solution quiescent. These modifications continue to be applied for the duration of the run. This option should not be used if the Vertical Equilibrium option is switched on (keyword **VE**).

- **THPRES** Enables the threshold pressure option.

  This prevents flow from occurring between different equilibration regions until the potential difference exceeds a threshold value. Threshold values must be specified with keyword **THPRES** in the SOLUTION section. This option switch is also required if named faults have threshold pressures (see keyword **THPRESFT**).

- **IRREVERS** The threshold pressures for flow in each direction between equilibration regions will be different. Threshold pressures must be specified separately for each direction. This is relevant only if the option **THPRES** is specified.

**DEFAULT:** None of the above

### Example

Enable the irreversible threshold option.

```plaintext
 EQLOPTS
   THPRES  IRREVERS /
```

**Keywords**

757
EQLZCORN  Reset part of the corner point depth array

The keyword may be followed by any number of records, each of which is terminated by a slash (/). The data is terminated by a null record (that is a record with no data before the terminating slash). Each record consists of at least one item of data.

1. The new value of the corner point (ZCORN) array.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   Each record may also contain the following items of data.
   Items 2-7 are used to redefine the input box for the modification to the corner depths. The input box usually only includes a single layer (KZ1=KZ2); a warning is issued if this is not the case.
   All the corner point depths for either the top or bottom (see item 12 below) of the layer will be set to the value specified in item 1.
   The modification is either continuous with the surrounding cells or discontinuous, depending on the settings of data items 8 to 11.
   If items 2-7 are not defined (a slash is inserted after item 1), they default to the values that were used for the previous operation within the current keyword. For the first operation in the keyword, the box defaults to the values set by the most recent BOX or ENDBOX keyword. If there is no preceding BOX or ENDBOX in the current section, the box is taken to include the entire reservoir.

2. IX1 First block to be modified on the X axis.
3. IX2 Last block to be modified on the X axis.
4. JY1 First block to be modified on the Y axis.
5. JY2 Last block to be modified on the Y axis.
6. KZ1 First block to be modified on the Z axis.
7. KZ2 Last block to be modified on the Z axis.
   The data must satisfy
   \[0 \leq IX1 \leq IX2 \leq NDIVIX\]
   \[0 \leq JY1 \leq JY2 \leq NDIVIY\]
   \[0 \leq KZ1 \leq KZ2 \leq NDIVIZ\] (where \(NDIVIX, NDIVIY\) and \(NDIVIZ\) are specified in RUNSPEC keyword DIMENS)
   However, if only certain individual corner depths within a block require to be set, any of \(IX1, IX2,\) or \(JY1, JY2\) may be zero (consider the possibilities in "Example 2”).
   Items 8-11 are used to specify whether the modification should be continuous with the surrounding cells.
8. \(IX1A = IX1-1\) or \(IX1\) (default = \(IX1-1\) if \(IX1 > 1\))
9. \(IX2A = IX2+1\) or \(IX2\) (default = \(IX2+1\) if \(IX2 < NDIVIX\))
10. \(JY1A = JY1-1\) or \(JY1\) (default = \(JY1-1\) if \(JY > 1\))
11. \(JY2A = JY2+1\) or \(JY2\) (default = \(JY2+1\) if \(JY2 < NDIVIY\) )
The data must satisfy

\[ 0 \leq IX1A \leq NDIVIX, 0 \leq IX2A \leq NDIVIX \]
\[ 0 \leq JY1A \leq NDIVIY, 0 \leq JY2A \leq NDIVIY \]

If, for example, \( IX2A = IX2 \), then blocks with \( I = IX2 + 1 \) are not modified and a discontinuity (or fault) is added to the surface. However, if \( IX2A = IX2 +1 \), then the corner points in blocks at \( I = IX2 +1 \) adjacent to the blocks at \( I = IX2 \) have the constant added, so preserving a continuous surface.

If the surface is discontinuous between \( I = IX2 \) and \( I = IX2 +1 \) prior to the action of the EQLZCORN keyword, then no action is taken with the blocks at \( I = IX2 +1 \).

By default items 8-11 are set to give a continuous surface.

12. Action required:

   **TOP**  Modify both top and bottom corner points of all cells, except for the bottom layer, in which the top cells only are modified

   **BOTTOM**  Modify both top and bottom corner points of all cells, except for the top layer, in which the bottom corners only are modified.

   (The word may be abbreviated; only the first character is significant.)

   **DEFAULT**: TOP

### Modifying selected corner point depths within a grid block

It is possible to modify selected corner depths within a grid block by specifying a zero in either the \( IX1 \), \( IX2 \) or \( JY1 \), \( JY2 \) box limits (or both). In this case only the corner depth on the appropriate side of the cell will be modified.

If data items 8-11 are defaulted or set to a box outside the box defined in data items 2-7, then the adjacent cell depths are modified to give a continuous surface. Note that if the surface was discontinuous prior to the EQLZCORN keyword, the adjacent cells are not modified.

See "Example 2".

See also the BOX, ENDBOX, and ADDZCORN keywords.

### Examples

#### Example 1

With \( NDIVIZ=5 \) specified in RUNSPEC keyword **DIMENS**:

```
EQLZCORN
4100.0   3  4  3  4   1  1          / Set the top of a region to 4100 ft without introducing faults.
4500.0  10 13  15 15  5  5  4* BOTTOM /Set the bottom surface/
```

#### Example 2

Set the top-left corner of cell (3,3,1) to 10890.0 ft
Set the top-right corner of cell (3,3). Modifying the top corner depth leaving the adjacent cells unmodified.

Set the lower-right corner of cell (3, 3, 5). Modifying the bottom corner depth leaving the adjacent cells unmodified.
EQUALREG  Sets an array to a constant value in a region

The keyword is followed by any number of records, each containing the items of data described below, and each terminated by a slash (/). It can be used to set an array to a constant value in a flux, MULTNUM or OPERNUM region.

The records can be terminated early using the slash (/) character, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

Each record consists of the following items of data:

1. The name of the array to be modified
2. The constant to be assigned to the array, specified by item 1
   The constant should be positive, and, for the REGIONS section, should be integer.
   DEFAULT: 0.0
3. The flux region number, MULTNUM region number, or OPERNUM region number (with choice of region determined using item 4)
   The flux region number refers to the region defined using the FLUXNUM keyword in the GRID section.
   It is not necessary to use the Flux Boundary option.
   The MULTNUM region number refers to the region defined using the MULTNUM keyword in the GRID section.
   The OPERNUM region number refers to the region defined using the OPERNUM keyword in the GRID or REGIONS section.
4. Choice of region: MULTNUM (M), FLUXNUM (F) or OPERNUM (O).
   DEFAULT: M

The set of records is ended by a blank record, containing only a slash (/).

Note: The FLUXNUM keyword, MULTNUM keyword or OPERNUM keyword must be input before the EQUALREG keyword. If item 4 is defaulted, then the MULTNUM keyword is used.

ECLIPSE 100
This keyword may not be used to modify the PBUB or PDEW arrays.

ECLIPSE 300
This keyword may be used in SCHEDULE section. However, this is a specialist facility.

Note: In the SCHEDULE section, regions (predefined by the FIPxxxx keyword in the REGIONS section) can be used in item 4.

Legal arrays

GRID section arrays

<table>
<thead>
<tr>
<th>DX</th>
<th>DY (DTHETA)</th>
<th>DZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>PERMX (PERMR)</td>
<td>PERMY (PERMHT)</td>
<td>PERMZ</td>
</tr>
</tbody>
</table>
MULTX (MULTR) MULTY (MULTTHT) MULTZ
PORO NTG DZNET
TOPS
DIFFMX (DIFFMR) DIFFMY (DIFFMTHT) DIFFMZ
FLUXNUM MULTNUM MPFANUM

ECLIPSE 300 only
MIDS

EDIT section arrays
PORV DEPTH
TRANX (TRANR) TRANY (TRANHT) TRANZ
DIFFX (DIFFR) DIFFY (DIFFHT) DIFFZ
HEATTX (HEATTR) HEATTY (HEATHTHT) HEATTZ (HEATTZ)
ROCKV

PROPS section arrays
SWL SWCR SWU
SGL SGCR SGU
KRW KRO KRG
PCG PCW

REGIONS section arrays
SATNUM PVTNUM EQLNUM
IMBNUM FIPNUM ENNUM
ROCKNUM MISCNUM

ECLIPSE 300 only
EOSUM

SOLUTION section arrays
PRESSURE SWAT SGAS
RV RS TBLK

ECLIPSE 100 only
GI OILAPI SALT
ECLIPSE 100 only
GASCONC SOLVCONC SOLVFRAC
ECLIPSE 100 only
SSOL SPOLY SURF

Keywords 762
ECLIPSE Reference Manual

ECLIPSE 100 only
SFOAM

ECLIPSE 300 only
SOIL   PBUB   PDEW

ECLIPSE 300 only
TEMP

SCHEDULE section arrays (specialist facility)

ECLIPSE 300 only
PORO   PERMX   PERMY

ECLIPSE 300 only
PERM2   MULTX   MULTY

ECLIPSE 300 only
MULT2   SIGMA(V) MULTSIG(V)

ECLIPSE 300 only
MULSGGD(V) DIFFMMF WORK(1/2)

Notes

- The above list is not exhaustive, but includes the most common examples.
- Keywords in brackets are for radial geometry runs.
- In the REGIONS section only integer values should be used.
- Each record must refer to the same region. It is not possible to choose a different region for each data record.
- The EQUALREG keyword is not available for use with local grid systems.

ECLIPSE 100 only
See also keywords ADDREG, FLUXNUM, MULTNUM, OPERNUM, MULTIREG, and COPYREG.

Additional notes

ECLIPSE 300 only
Additional notes for the use of this keyword in the SCHEDULE section. Again, this is a specialist facility. In the SCHEDULE section, the effect of this keyword will be as follows:

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DIFFMMF</td>
<td>Modifies the matrix-fracture diffusivity in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as DIFFMMFS.</td>
</tr>
<tr>
<td>MULSGGD</td>
<td>Modifies the matrix-fracture coupling transmissibility for oil-gas gravity drainage in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as MULSGGDS.</td>
</tr>
<tr>
<td>MULSGGDV</td>
<td>Modifies the matrix-fracture coupling transmissibility in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as MULTSIGS.</td>
</tr>
<tr>
<td>MULTSIG</td>
<td>Modifies the matrix-fracture coupling transmissibility in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as MULTSIGS.</td>
</tr>
<tr>
<td>MULTX</td>
<td>Modifies transmissibility in the X/Y/Z-direction. If output is required for flexible restart in the model, the multiplier values will be output as MULTX/MULTY/MULTZ.</td>
</tr>
</tbody>
</table>

Keywords
<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PERMX</td>
<td>Modifies permeability in the X/Y/Z-direction. Also approximates the corresponding transmissibility multiplier MULTX/MULTY/MULTZ determined from the ratio of the new permeability value to the old one. If output is required for flexible restart in the model, the permeability values will be output as MODPERMX / MODPERMY / MODPERMZ.</td>
</tr>
<tr>
<td>PERMY</td>
<td></td>
</tr>
<tr>
<td>PERMZ</td>
<td></td>
</tr>
<tr>
<td>PORO</td>
<td>Modifies porosity. Modifies the corresponding pore volume by a multiplier MULTPV determined from the ratio of the new porosity value to the old one. If output is required for flexible restart in the model, the porosity values will be output as MODPORO, and the pore volume multipliers will be output as MULTPVs.</td>
</tr>
<tr>
<td>SIGMA</td>
<td></td>
</tr>
<tr>
<td>SIGMAV</td>
<td></td>
</tr>
<tr>
<td>WORK(1/2)</td>
<td>These are work arrays to be stored for subsequent use. If output is required for flexible restart in the model, these values will be output as WORK(1/2).</td>
</tr>
</tbody>
</table>

Table 3.14: Additional notes for EQUALREG in the SCHEDULE section

Examples

Example 1

NX=6, NY=8, NZ=1:

```plaintext
FLUXNUM
1 1 2 2 2 2
1 1 2 2 2
4 4 1 1 3 3
4 4 4 1 3 3
4 4 4 1 3 3
5 5 5 5 5 5
4 4 1 1 3 3
4 4 4 1 3 3 /
EQUALREG
PORO 0.1 1 F /
PORO 0.15 2 F /
PORO 0.13 4 F /
PORO 0.11 3 F /
PORO 0.3 5 F /
/
```

Example 2

NX=6, NY=2, NZ=1:

```plaintext
MULTNUM
1 1 2 2 2 2
1 1 2 2 2 2 /
EQUALREG
PERMX 0.1 1 /
PERMX 0.15 2 /
/
```
Set array to a constant in current box

This keyword is used to assign or replace the value of a property for a box of cells within the grid.

The keyword may be followed by any number of records, each of which is terminated by a slash (/). The data is terminated by a null record (that is, a record with no data before the terminating slash(/)).

Each record consists of at least two items of data:

1. The name of the array to be modified
2. The constant to be assigned to the array specified by item 1

   The constant should be positive, and must be an integer if the array contains integer region numbers (for example, FIPNUM). The constant may be real if the array contains real data (for example, TOPS or TRANX). This item should not be defaulted.

   The following items of data may be used to redefine the input box for this and subsequent operations within the current keyword.

   If these items are not defined (a slash is inserted after item 2), they default to the values which were used for the previous operation within the current keyword.

   For the first operation in the keyword, the box defaults to the values set by the most recent BOX or ENDBOX keyword. If there is no preceding BOX or ENDBOX in the current section, the box is taken to include the entire reservoir.

3. First block to be modified on the X axis (IX1)
4. Last block to be modified on the X axis (IX2)
5. First block to be modified on the Y axis (JY1)
6. Last block to be modified on the Y axis (JY2)
7. First block to be modified on the Z axis (KZ1)
8. Last block to be modified on the Z axis (KZ2)

The data must satisfy:

\[ 1 \leq \text{IX1} \leq \text{IX2} \leq \text{NDX} \]
\[ 1 \leq \text{JY1} \leq \text{JY2} \leq \text{NDY} \]
\[ 1 \leq \text{KZ1} \leq \text{KZ2} \leq \text{NDZ}, \]

where NDX, NDY, NDZ are the limits in the current BOX.

See also keywords BOX, ENDBOX, ADD, MULTIPLY and COPY.

Legal arrays

GRID section arrays

<table>
<thead>
<tr>
<th>DX (DR)</th>
<th>DY (DTHETA)</th>
<th>DZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>PERMX (PERMR)</td>
<td>PERMY (PERMTHT)</td>
<td>PERMZ</td>
</tr>
<tr>
<td>MULTX (MULTR)</td>
<td>MULTY (MULTTHT)</td>
<td>MULTZ</td>
</tr>
</tbody>
</table>
ECLIPSE Reference Manual

PORO     NTG     DZNET
TOPS
DIFFMX (DIFFMR) DIFFMY (DIFFMTHT) DIFFMZ
FLUXNUM   MULTNUM  MPANUM

ECLIPSE 100 only
MULTPV

ECLIPSE 300 only
MIDS    PERMMF

EDIT section arrays
PORV     DEPTH
TRANX (TRANR) TRANY (TRANHT) TRANZ
DIFFX (DIFFR) DIFFY (DIFFHT) DIFFZ

ECLIPSE 100 only
MULTPV

ECLIPSE Thermal only
HEATTX (HEATTR) HEATTY (HEATTHT) HEATTZ (HEATTZ)

ECLIPSE Thermal only
ROCKV

PROPS section arrays
SWL     SWCR    SWU
SGL     SGCR    SGU
KRW     KRO     KRG
PCG     PCW

REGIONS section arrays
SATNUM   PVNUM   EQNUM
IMBNUM   FIPNUM  ENDNUM
ROCKNUM  MISCNUM SURFNUM

ECLIPSE 300 only
EOSNUM

SOLUTION section arrays
PRESSURE SWAT   SGAS
RV       RS      TBLK

ECLIPSE 100 only
GI       OILAPI  SALT

ECLIPSE 100 only
GASCONC  SOLVCONC SOLVFRAC

Keywords

766
Notes

- The above list is not exhaustive, but includes the most common examples.
- Keywords in brackets are for radial geometry runs
- In the REGIONS section only integer values should be used
- The ADD, COPY, EQUALS and MULTIPLY keywords are processed as they are read, so that repeated operations are possible
- For arrays marked with an asterisk, EQUALS may only be used to replace existing values, not assign the array from scratch.
- **Item 2 should not be defaulted.** However, this facility is retained for back-compatibility. If this item is defaulted, this will be treated differently by ECLIPSE 100 and ECLIPSE 300. For ECLIPSE 100, the defaulted value is assumed to be undefined, whereas for ECLIPSE 300, the defaulted value is assumed to be zero. The subsequent processing will depend, therefore, both upon the specific keyword to which the operation is being applied and which simulator is being used.

Examples

Example 1

In the GRID section:

```
------- ARRAY  CONSTANT  ----- BOX ----- 
EQUALS  
  DX  1000  1 11 1 19 1 1 /
  DY  1000  / defaults to last specified box
  DZ   10  / defaults to last specified box
  PORO  0.19  / defaults to last specified box
  PERMX 230  / defaults to last specified box
  TOPS 6845  / defaults to last specified box
  TOPS 6872  1 11 1 19 2 2 /
  TOPS 6901  1 11 1 19 3 3 /
  TOPS 6933  1 11 1 19 4 4 /
/
```

Example 2

In the EDIT section:

```
------- ARRAY  CONSTANT  ----- BOX ----- 
EQUALS  
  TRANY 1000  1 11 1 19 1 1 /
  FORV 1000  / defaults to last specified box
/
```
Equilibration data specification

The keyword sets the contacts and pressures for conventional hydrostatic equilibrium. It should be followed by NTEQUL records of data, each record terminated with a slash (/). NTEQUL refers to the number of equilibration regions, as entered in the RUNSPEC section (keyword EQLDIMS).

For non-Thermal simulations, this keyword should not be used for non-equilibrium initialization runs, and precludes the use of PBUB, PDEW, SWAT, SGAS, SOIL, XMF and YMF for example. For more information, refer to "Initializing the study" in the ECLIPSE Technical Description.

Note: An additional initialization method has been added for Thermal simulations. This mixed hydrostatic equilibration option should be triggered using only items 1 and 2 of the EQUIL keyword, together with saturation arrays (SOIL, for instance) and the NEI keyword. For more information, refer to "Initializing the study" in the ECLIPSE Technical Description. This initialization method should not be mixed with the standard equilibration method, otherwise an error message will be issued.

Each record refers to a separate equilibration region (1 to NTEQUL), and contains the following items of data:

1. Datum depth
   In problems containing dissolved gas or vaporized oil, if RS or RV versus depth tables are not provided (see items 7 and 8), the datum depth must lie at the gas-oil contact.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

   ECLIPSE 100 only
   ECLIPSE 300 only
   DEFAULT: 0.0
   Default not defined

2. Pressure at the datum depth.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

   Note: If the datum depth lies above the gas-oil contact, the pressure refers to the gas phase. If the datum depth lies below the water-oil contact, the pressure refers to the water phase. Otherwise, the pressure refers to the oil phase.

3. Contact depth
   Depth of the water-oil contact (for three-phase or oil-water problems) or
   Depth of the gas-water contact (for gas-water problems).
   This value is ignored in single-phase and oil-gas problems.
   The contact can lie below the bottom of the reservoir if there is no mobile water initially present.
   The values of the water saturation above and below the transition region are set at the end-point saturations in the water capillary pressure table.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   DEFAULT: 0.0
4. Capillary pressure at the contact

Oil-water capillary pressure at the water-oil contact (for three-phase or oil-water problems) or
Gas-water capillary pressure at the gas-water contact (for gas-water problems).
This value is ignored in single-phase and oil-gas problems.
Capillary pressures rather than saturations are used in order that more than one saturation table may be
used in an equilibration region.

UNITs: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
DEFaulT: 0.0

5. Depth of the gas-oil contact.

This value is ignored in single-phase, oil-water and gas-water problems.

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
DEFaulT: 0.0

6. Gas-oil capillary pressure at the gas-oil contact

This value is ignored in single-phase, oil-water and gas-water problems.

UNITs: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
DEFaulT: 0.0

7. Integer selecting the type of initialization for live black oil

A positive integer causes the dissolved gas concentration in undersaturated oil to be calculated from
either an $R_s$ versus depth table or a $P_b$ versus depth table, subject to an upper limit equal to the
saturated $R_s$ value at the local pressure. The $R_s$ versus Depth table is entered using keyword $RSVD$.
The $P_b$ versus Depth table is entered using keyword $PBVD$. If multiple tables are defined, the table
applicable to a grid cell is that specified by the $EQLNUM$ keyword, which defines equilibration regions.

A zero or negative value causes the dissolved gas concentration in undersaturated oil to be set equal to
the saturated $R_s$ value at the gas-oil contact. In this case, an $R_s$ versus depth or a $P_b$ versus depth table
need not be provided, but the datum depth (item 1) must coincide with the gas-oil contact, that is the
bubble point is set to the local pressure at the reference depth.)
This value is ignored if dissolved gas is not present in black oil runs.
This value is ignored in compositional runs.

DEFAULT: 0

Black Oil cases only

8. Integer selecting the type of initialization for black oil runs with wet gas

A positive integer causes the vaporized oil concentration in undersaturated gas to be calculated from either an \( R_v \) versus depth table or a \( P_d \) versus Depth table, subject to an upper limit equal to the saturated \( R_v \) value at the local pressure. The \( R_v \) versus depth table is entered using keyword RVVD. The \( P_d \) versus depth table is entered using keyword PDVD. If multiple tables are defined, the table applicable to a grid cell is that specified by the EQLNUM keyword, which defines equilibration regions.

A zero or negative value causes the vaporized oil concentration in undersaturated gas to be set equal to the saturated \( R_v \) value at the gas-oil contact, subject to an upper limit equal to the saturated \( R_v \) value at the local pressure. In this case, an \( R_v \) versus Depth or a \( P_d \) versus depth table need not be provided, but the datum depth (Item 1) must coincide with the gas-oil contact (that is the dew point is set to the local pressure at the reference depth).

This value is ignored if vaporized oil is not present in black oil runs.
This value is ignored in compositional runs.

DEFAULT: 0

9. Integer (N) defining the accuracy of the initial fluids in place calculation.

\( N=0 \) Causes the simulator to set the fluid saturations in each grid block according to the conditions at the center of the block. This produces a steady-state solution, but the fluids in place will not be accurate if a contact passes near the center of a large grid block.

\( N<0 \) Causes the simulator to take an average of the conditions at \( 2|N| + 1 \) equally-spaced sub-blocks within each grid block. The blocks are treated as being horizontal. The larger \( N \) is, the more accurate will be the initial fluids in place calculation. But the resulting initial conditions will not be absolutely steady-state, in that there will be a small readjustment if the simulator is run for a period with no wells.

\( N>0 \) Is equivalent to the \( N < 0 \) option, but the grid blocks are treated as being tilted. If the NEWTRAN method has been used, the tilt on the top and bottom faces of each grid block is determined from the block’s corner point depths. Otherwise the top and bottom faces are orientated towards their neighbors within the equilibration region. The simulator takes an average of the conditions at \( N \) levels within each half (upper and lower) of the block, weighted according to the block’s horizontal cross-section at each level.

This option cannot be used in radial geometry.

\( N \) is limited to 20.

---

**Note:** When \( N \) is non-zero, if the QUIESCENCE option is selected (using keyword EQLOPTS), modifications will be applied to the phase pressures to make the initial solution a true steady-state. These pressure modifications will be applied for the duration of the run. The absolute size of \( N \) is limited to 20.
The QUIESCENCE option should not be selected when \( N=0 \) as the initial conditions will always be quiescent in this case.
**ECLIPSE 300**

*Note:* For back compatibility with pre-2000A versions of ECLIPSE 300, the OLDEQUIL will cause ECLIPSE 300 to use horizontal block integration in equally spaced N layers.

**ECLIPSE 100**

*Note:* If the vertical equilibrium option is being used, however, (keyword `VE`), an exact VE equilibration method will be used if N = 0. The fluid saturations are determined directly from the depth of the contact levels within each block. This is the preferred option for VE runs. Refer to the `VEFIN` keyword and see also “Vertical equilibrium” in the *ECLIPSE Technical Description* for details of the behavior in mixed VE/non-VE LGR runs.

DEFAULT: In ECLIPSE 100, the value is defaulted to -5 (giving a 10-point average of conditions within each block), unless the vertical equilibrium option is being used, in which case it is defaulted to 0. In ECLIPSE 300, the value is defaulted to 0.

**ECLIPSE 300**

10. Integer selecting the type of initialization in the compositional case.

For a gas-water run, the data is entered as for an oil-water system - that is the contact depth and capillary pressure as items 3 and 4.

See "Initializing the study" in the *ECLIPSE Technical Description* for more details of the initialization of compositional studies.

Possible values are:

1. Continuous hydrocarbon phase initial state (that is no gas-oil contact in the reservoir). This includes super critical fluids which make a smooth transition from a gas to an oil.

2. Gas-oil contact, with vapor composition specified at contact. Pressure at datum depth will be set to retrograde dew point pressure

3. Gas-oil contact, with liquid composition specified at contact. Pressure at datum depth is set to bubble point pressure

DEFAULT: 1.

*Note:* If item 10 is set to 1 and a gas-oil contact is specified within the reservoir, then to ensure proper phase-labeling there must be compositional variation across the contact. The compositional variation can be specified using `COMPVD` or `ZMFVD`. This requirement is implemented in 2009.1. For the pre-2009.1 calculation, item 172 of `OPTIONS3` should be set to a non-zero value.

**ECLIPSE 300**

11. If this is set to 1 the field pressure is not set to the saturation pressure at the contact if options 2 or 3 are selected in item 10.

*Note:* If item 10 is set to 2 or 3 in the compositional case, it is assumed that the datum depth corresponds to the gas-oil contact. Given the vapor composition at the contact (if item 10 = 2) or the oil compositional at the contact (if item 10 = 3), then the dew point or bubble point pressure will be calculated. This saturation pressure, $P_{sat}$, should be equal to the datum pressure, $P_{datum}$, specified in item 2 of this keyword. If item 11 is not set, then the test $|P_{datum} - P_{sat}| < \varepsilon$ where $\varepsilon$ is 1 atmosphere is performed. If the condition is not true, the datum pressure is reset to the calculated saturation pressure. Setting item 11 = 1 ensures the user specified datum pressure is retained regardless of the result of the
above test; note this is not an equilibrium system. See "Initializing the study" in the ECLIPSE Technical Description for more details.

Notes

The end points of the water and gas saturation tables (see SWFN and SGFN) determine the phase saturations outside the transition zones. It is therefore important that these should be consistent with each other. The highest gas saturation value \( S_{g,max} \) in the gas table should not exceed one minus the lowest water saturation value \( S_{w,min} \) in the water saturation table, otherwise a negative oil saturation would result in the gas cap. If there is to be no oil in the gas cap above the gas-oil transition zone, then set

\[
S_{g,max} = 1 - S_{w,min}
\]

Eq. 3.69

A residual oil saturation \( S_{o,res} \) in the gas cap can be produced by setting

\[
S_{g,max} = 1 - S_{w,min} - S_{o,res}
\]

Eq. 3.70

In problems containing a gas-water contact, the water saturation tables are assumed to contain the water-gas capillary pressure values versus water saturation. The gas capillary pressure values should be set to zero.

The equilibration calculation assumes that gas is lighter than oil and oil is lighter than water at reservoir conditions. An error condition may result if this is not so.

ECLIPSE 100 only

It is also possible to specify negative depth values.

ECLIPSE 300 only

By default, oil and gas phase pressures are derived from the pressure solution variable \( P \) using the formula

\[
Poil = P - P_{cog} \times \left( \frac{S_g}{S_o + S_g} \right)
\]

and

\[
P_{gas} = P + P_{cog} \times \left( \frac{S_o}{S_o + S_g} \right)
\]

where

\( P_{cog} \) is the gas-oil capillary pressure,

\( S_o \) and \( S_g \) are the oil and gas saturations respectively.

This Pc-splitting technique is performed to enable better convergence and property smoothness above the critical point. It may be disabled using keyword FORMOPTS, option NOPCSPT. When reporting initialization data using the RPTST, RPTSol or OUTSOL keywords, the mnemonic PRESSURE is the solution variable \( P \) which is derived from the phase pressures calculated during the EQUIL hydrostatic equilibration and reported with the SOLVD option in RPTSol, POil is the oil phase pressure \( P_{oil} \) and PGAS is the gas phase pressure \( P_{gas} \), as discussed in the above two equations.

Note: The Pc-splitting technique applies only to the ECLIPSE 300 compositional mode.
Example

With NTEQUIL=2:

```
EQUIL
  7100 4000 8000 0.0 6500 0.0 1 0 -3   /
  7000 4000 7800 0.0 7000 0.0          /
```
Salt concentration nodes for polymer viscosity multiplier

The data comprises NTPVT (see RUNSPEC keyword TABDIMS) tables of effective salinity values, each terminated by a slash (/), describing the nodal values to be used in the calculation of polymer viscosity multipliers.

Each table consists of up to NPPVT (see RUNSPEC keyword TABDIMS) values of the effective salinity calculated as specified by the SALTEFF keyword. Within each table the values should be strictly increasing.

The number of entries in each table must correspond to the number of viscosity multiplier entries used in the PLYVISC keyword.

See "Polymer flood model" in the ECLIPSE Technical Description for further information.

Example

With NTPVT=2 and NPPVT≥3:

```
ESPNODE
  0
  50
  100/
  0
  15
  35/
```

Units

Depending on the coefficients provided in the SALTEFF keyword, effective salinity can have the following units (see "Effective salinity calculations" in the ECLIPSE Technical Description):

- UNITS: kg/sm³ (METRIC), lb/stb (FIELD), gm/scf (LAB), or
- UNITS: sm³/kg (METRIC), stb/lb (FIELD), scc/gm (LAB), or
- UNITS: dimensionless.
ESSNODE  Salt concentration nodes for water/oil surface tension

The data comprises NTPVT (see RUNSPEC keyword TABDIMS) tables of salt concentration (or effective salinity) values, each terminated by a slash (/), describing the nodal values for salt concentration (or effective salinity) to be used in the calculation of the oil-water surface tension.

Each table consists of up to NPPVT (see RUNSPEC keyword TABDIMS) values of the salt concentration. Within each table the values should be strictly increasing.

The number of entries in each table must correspond to the number of surface tension entries used in the SURFSTES keyword.

In ECLIPSE 100, the values of salt concentration should be non-negative. If keyword ECLMC is selected, the BRINE keyword activates the Multi-Component Brine Model which can be used with SURFSTES and ESSNODE.

In ECLIPSE 300, the effective salinity of the water phase is computed as a function of the water component concentrations. The exact expression for this computation can be specified by keyword SALTEFF.

Example

With NTPVT=2 and NPPVT≥3:

```
ESSNODE
0
50
100/
0
15
35/
```

Units

**ECLIPSE 100 only**

Depending on whether salinity-dependent surfactant model is activated together with brine or multi-component brine options, ESSNODE will contain values of salt concentration or effective salinity respectively. Accordingly, the units will be

UNITS: kg/sm^3 (METRIC), lb/stb (FIELD), gm/scc (LAB), or

UNITS: dimensionless.

**ECLIPSE 300 only**

Depending on the coefficients provided in the SALTEFF keyword, effective salinity can have the following units (see "Effective salinity calculations" in the ECLIPSE Technical Description):

UNITS: kg/sm^3 (METRIC), lb/stb (FIELD), gm/scc (LAB), or

UNITS: dimensionless.
EXCAVATE

Remove grid blocks during a run

The keyword should be followed by one integer for every grid block in the current input box. The data
must be terminated by a slash (/).

The values indicate whether the corresponding grid block is to be excavated. A value of 0 indicates the
block is to remain and a value of 1 indicates the block should be removed.

Excavated grid blocks are technically still present in the simulation but have transmissibilities of zero in all
directions resulting in no communication with the rest of the reservoir. An aquifer may be attached to the
edge of an excavated region to model boundary conditions. Note that AQUCON item 11 must be set to YES
in order to do this.

Any well or aquifer connections in excavated blocks will be automatically shut. Blocks containing river
connections cannot be excavated.

EXCAVATE can be placed between a REFINE / ENDFIN pair, enabling the excavation of cells in a local
grid refinement. If a global cell containing a local grid refinement is excavated, the corresponding local
grid cells will be excavated as well.

Fluid-in-place calculations and pressure averages (such as field pressure) do not include contributions from
excavated blocks unless OPTIONS item 146 is set to a non-zero value.

The excavation status of a grid block can be visualized as a 3D property using the identifier EXCAVNUM.

EXCAVATE is not permitted in dual porosity runs.

Example

```
BOX
2 4 5 10 1 1 /
EXCAVATE
0 1 0 12*1 0 1 0 /
ENDBOX
```
Requests run summary output to be in Excel format

This keyword requests that the run summary output, generated by using the RUNSUM keyword, should be written in a format that can be easily imported into Excel.

In ECLIPSE 100, if this keyword is used the run summaries are output to a separate file rather than appended to the end of the Print file. (There is no need to use the keyword SEPARATE with EXCEL.) The separate run summary file name is the root with the extension RSM.

In ECLIPSE 300, the run summary file name is the root with the extension RSM. This separate file is automatically generated.

Columns of data are tab-delimited. The file can be imported into Excel by using the File | Open option from within the spreadsheet. No further commands are required; the import wizard can be finished immediately without any further steps.

The EXCEL keyword has no associated data.

Example

```
EXCEL
```
EXTFIN

Specifies an external local grid refinement

This keyword must be used to specify local grid refinements when using an unstructured grid from an external source, typically FloGrid. It is not intended for general use. The keywords EXTHOST and TRANGL must also be specified, and EXTREPGL may also be specified if required.

The keyword is followed by the following items, terminated with a slash (/).

1. Name of the local grid refinement (up to 8 characters).
2. NX I-dimension of the refinement.
3. NY J-dimension of the refinement.
4. NZ K-dimension of the refinement.
5. NREPG Number of replaced global cells in the refinement.

ECLIPSE 100 only
6. NHALO Number of global halo cells for the refinement (this is the number of cells in the global grid that connect to cells in the local grid using non-neighbor connections across the local grid boundary).

ECLIPSE 100 only
7. NFLOG Maximum number of global flows into or within the refinement.

ECLIPSE 100 only
8. NUMINT Maximum number of internal local flows in the refinement crossing boundaries between global cells.

ECLIPSE 100 only
9. NUMCON Maximum number of global-local connections.

ECLIPSE 100 only
10. NWMAX Maximum number of wells in the refinement.

Any other GRID section keywords (for example PORO and EQUALS) following this are taken to apply to this local refined grid, until ENDFIN or another REFIN, EXTFIN, CARFIN, RADFIN or RADFIN4 keyword is encountered.

ECLIPSE 300
Any data entered for items 6-10 is ignored.

Example

```
EXTFIN
  -- NAME    NX  NY  NZ NHOST NHALO NFLOG NUMINT NUMCON NWMAX
  EAST    12  10  6  15  9  10  65  34  2 /
```

Keywords

778
**EXTHOST**

**Specify parent cells for LGR cells in an external local grid refinement**

This keyword is obligatory for local grids defined using keyword `EXTFIN` for an unstructured host grid, being typically supplied by FloGrid. It specifies the global host cells in natural cell ordering. See also keywords `EXTFIN`, `EXTREPGL` and `TRANGL`.

The keyword should be followed by an integer for every cell in the current box. A value equal to or less than zero signifies that the local cell is just a fill-in in the $NX \times NY \times NZ$ mapping used for the unstructured local grid.

Host cell data must be input for every cell in the external local grid. Grid blocks are ordered with the I-direction cycling fastest, followed by the J and then the K indices. Repeat counts may be used for repeated values.

**Example**

```
EXTHOST
--- NX*NY*NZ=8
  2 4 5 2 4 5 2 5 /
```
EXTRAPMS Requests warning messages for table extrapolation

The keyword should be followed by a line containing a single integer, terminated with a slash (/). The integer requests that warning messages should be printed at the end of a timestep in which the PVT or VFP tables have been extrapolated.

0  No warnings given
1  Warnings given for PVT table extrapolation
2  Warnings given for VFP table extrapolation
3  Warnings given for PVT and VFP table extrapolation
4  As for 3, but report additional diagnostics to the Print file

R_s and R_v extrapolation warnings will be given for values 1 and 3.

If this keyword is omitted, no extrapolation warnings will be given.

Example

```
EXTRAPMS
  3 /
```
**EXTREPG** Specify replaced global cells for an external local grid refinement

This keyword may be used for local grids defined using keyword EXTFIN for an unstructured local grid in a structured host grid, being typically supplied by FloGrid. It specifies the global cells which are to be replaced by cells in the unstructured LGR in natural cell ordering. See also keywords EXTFIN, EXTHOST and TRANGL.

The keyword should be followed by NREPG integers, where NREPG is the number of replaced global cells specified in item 5 of the EXTFIN keyword.

Grid blocks in the parent structured grid are ordered with the I-direction cycling fastest, followed by the J and then the K indices. Repeat counts may be used for repeated values.

If this keyword is not specified for an EXTFIN refinement, the replaced global cells are assumed to be identified wholly by the host cells specified using EXTHOST.

**Example**

```
EXTREPG
    -- NHOST=5
    2 5 9 4 15 /
```
This section contains an alphabetic listing of keywords starting with the letter F.
**FACTLI**

**Modify Li critical temperature correlation**

The keyword modifies the Li critical temperature correlation. This is normally set up automatically, but the FACTLI keyword allows this to be modified.

A value is required for each equilibration region.

**DEFAULT: 1.0**

The Li correlation for the critical temperature of a fluid mixture is:

\[
T_{\text{crit mix}} = \frac{\sum j T_{cj} \cdot V_{cj} \cdot z_j}{\sum j V_{cj} z_j}
\]

Eq. 3.71

For single phase cells for which no bubble or dew point fluid can be found, this critical temperature is compared with the reservoir temperature to establish if the fluid is an oil or a gas. This is usually only required well away from the two phase region.

The item in FACTLI adds a multiplying factor to \( T_{\text{crit mix}} \). This may be used to tune the point at which a cell is nominally deemed to be oil or gas. For example, for a rich condensate, a value less than 1 might be entered to prevent states being taken as oil.

Typical FACTLI values lie between 0.7 and 1.3.

**Note:** If compositional equilibration option 1 is selected in item 10 of keyword EQUIL (single phase system) and the GOC is set within the reservoir, then the FACTLI value will be set so that states above the GOC are labelled as gas and those below as oil. This will only occur when compositional variation with depth exists and is entered using ZMFVD or COMFVD.

**Examples**

**Example 1**

If the reservoir is known to be a gas, but single phase oil cells are being predicted, the Li estimate of the critical point is modified by 0.8 to lower it so that one phase cells are more inclined to be reported as gas:

```
FACTLI 0.8 /
```

**Example 2**

Example with two equilibration regions:

```
FACTLI 0.98 0.87 /
```

**Keywords**

783
FASTTRAC  

Enable shortcuts in tracer calculations

This indicates that no checks are performed on the tracer solution to ensure convergence. There is no recalculation of the residual of the tracer equations and there is only the one update (one Newton iteration).

Enabling this option can save substantial amounts of time when large numbers of tracers are present, for example when using the TRACK or delumping options.

**CAUTION:** However, it does mean there is no control to ensure that the residuals do meet the (stringent) convergence tests of the tracer tracking code. This could possibly lead to differences between results with and without this keyword.

The keyword has no associated data.
**FAULTDIM**  

**Dimensions for fault data**

This keyword specifies the maximum number of segments of fault data entered with the `FAULTS` keyword in the `GRID` section.

The keyword should be followed by a single item of data, terminated by a slash (/).

1. **MFSEGS** The maximum number of fault segments
   
   DEFAULT: 0

**Example**

```plaintext
FAULTDIM
3 /
```
FAULTS

Specifies faults for later editing

This keyword is used to define a set of faults, which can subsequently have their transmissibilities (and diffusivities) modified using the keyword MULTFLT.

The FAULTS keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/):

1. Fault name (up to 8 characters)
   The fault is made up of all the segments with the same name. The segments may be placed in any order.

2. Lower I-coordinate of cells along the fault (IX1)

3. Upper I-coordinate of cells along the fault (IX2)
   IX1 must equal IX2 if the face (item 8) is X or I

4. Lower J-coordinate of cells along the fault (IY1)

5. Upper J-coordinate of cells along the fault (IY2)
   IY1 must equal IY2 if the face (item 8) is Y or J

6. Lower K-coordinate of cells along the fault (IZ1)

7. Upper K-coordinate of cells along the fault (IZ2)
   IZ1 must equal IZ if the face (item 8) is Z or K

8. Face of the fault
   This should be one of X, Y, Z or I, J, K.

   If negative direction transmissibility multipliers, MULTX-, MULTY-, or MULTZ- are enabled (using keyword GRIDOPTS in the RUNSPEC section), then the face may also be one of X-, Y-, Z- or I-, J-, K-.

The set of records must end with a blank record, containing only a slash (/).

Notes

• The FAULTS keyword may be used more than once within the GRID section.

   Recall that the FAULTDIM keyword in the RUNSPEC section is used to specify the maximum number of fault segments that can be entered using the FAULTS keyword. Therefore, for this simulator, the FAULTS keyword may be used more than once within the GRID section, provided that the total number of records does not exceed the value specified in FAULTDIM.

• The faults defined in this keyword do not have to correspond to the actual displacement faults in the corner-point geometry (keywords COORD and ZCORN). The faults defined here are only used as a convenient way of varying the transmissibility along the fault trajectory with the MULTFLT keyword. The generation of the non-neighbor connections is still governed by the corner-point geometry, and does not depend on the fault definitions in the FAULTS keyword (except that keyword MULTFLT modifies any non-neighbor connection transmissibilities across named fault segments).

• The fault data can be printed out by including the mnemonic FAULTS in the RPTGRID keyword.
Example

```
Example

<table>
<thead>
<tr>
<th>FAULTS</th>
<th>IX1</th>
<th>IX2</th>
<th>IY1</th>
<th>IY2</th>
<th>IZ1</th>
<th>IZ2</th>
<th>FACE</th>
</tr>
</thead>
<tbody>
<tr>
<td>'zigzag'</td>
<td>1</td>
<td>1</td>
<td>2</td>
<td>1</td>
<td>4</td>
<td>X</td>
<td>/</td>
</tr>
<tr>
<td>'zigzag'</td>
<td>2</td>
<td>2</td>
<td>2</td>
<td>1</td>
<td>4</td>
<td>Y</td>
<td>/</td>
</tr>
<tr>
<td>'zigzag'</td>
<td>2</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>1</td>
<td>4</td>
<td>X</td>
</tr>
<tr>
<td>'zigzag'</td>
<td>3</td>
<td>3</td>
<td>4</td>
<td>4</td>
<td>1</td>
<td>4</td>
<td>Y</td>
</tr>
<tr>
<td>'zigzag'</td>
<td>3</td>
<td>3</td>
<td>5</td>
<td>6</td>
<td>1</td>
<td>4</td>
<td>X</td>
</tr>
<tr>
<td>'zigzag'</td>
<td>4</td>
<td>4</td>
<td>6</td>
<td>6</td>
<td>1</td>
<td>4</td>
<td>Y</td>
</tr>
<tr>
<td>'zigzag'</td>
<td>4</td>
<td>4</td>
<td>7</td>
<td>8</td>
<td>1</td>
<td>4</td>
<td>X</td>
</tr>
<tr>
<td>'zigzag'</td>
<td>5</td>
<td>5</td>
<td>8</td>
<td>8</td>
<td>1</td>
<td>4</td>
<td>Y</td>
</tr>
<tr>
<td>'zigzag'</td>
<td>5</td>
<td>5</td>
<td>9</td>
<td>10</td>
<td>1</td>
<td>4</td>
<td>X</td>
</tr>
<tr>
<td>'zigzag'</td>
<td>6</td>
<td>6</td>
<td>10</td>
<td>10</td>
<td>1</td>
<td>4</td>
<td>Y</td>
</tr>
<tr>
<td>'block'</td>
<td>19</td>
<td>19</td>
<td>20</td>
<td>30</td>
<td>1</td>
<td>4</td>
<td>X</td>
</tr>
<tr>
<td>'block'</td>
<td>30</td>
<td>30</td>
<td>20</td>
<td>30</td>
<td>1</td>
<td>4</td>
<td>X</td>
</tr>
<tr>
<td>'block'</td>
<td>20</td>
<td>30</td>
<td>19</td>
<td>19</td>
<td>1</td>
<td>4</td>
<td>Y</td>
</tr>
<tr>
<td>'block'</td>
<td>20</td>
<td>30</td>
<td>30</td>
<td>30</td>
<td>1</td>
<td>4</td>
<td>Y</td>
</tr>
</tbody>
</table>
```

For the above example, a value of 14 has been specified in the `FAULTDIM` keyword in the `RUNSPEC` section.

ECLIPSE 100

For the above example, a value of 14 has been specified in the `FAULTDIM` keyword in the `RUNSPEC` section.
Flash convergence criteria

This is an optional keyword which modifies the flash convergence criteria used in a compositional run. It may be used in the SOLUTION and SCHEDULE sections and will carry over from the SOLUTION section; moreover, it may be used several times in the SCHEDULE section if needed.

1. Maximum number of iterations in the vector flash calculation.
   DEFAULT: 6
2. Vector flash error target for explicit cells.
   DEFAULT: 1.0E-3
3. Vector flash error target for implicit cells.
   DEFAULT: 1.0E-6
4. Vector flash error target for reservoir initialization. This parameter only influences the accuracy at the beginning of the simulation and is superseded by items 2 or 3 for explicit and implicit cells respectively.
   DEFAULT: 1.0E-8
5. Maximum number of successive-substitution iterations for the stability check.
   DEFAULT: 12
6. Error tolerance for the successive-substitution phase of the stability check
   DEFAULT: 1.0E-8
7. Maximum number of BFGS (second order) iterations for the stability check.
   DEFAULT: 12
8. Error tolerance for the BFGS (second order) phase of the stability check.
   DEFAULT: 1.0E-8

Example

```
FCCRIT
8 1.0E-4 1.0E-7 /
```
FHERCHBL  Define property data for a Herschel-Bulkley fluid as a function of polymer concentration

This keyword provides information that is used to modify the partially mixed water viscosity in the Polymer Flood. Therefore this keyword has to be used in conjunction with the polymer flood option (refer to the POLYMER keyword).

The reference viscosity is the water viscosity in the Polymer Flood Model. This keyword must be used in conjunction with the NNEWTF keyword in the RUNSPEC section. For more details please refer to the "Non-Newtonian fluid flows" option in the ECLIPSE Technical Description.

The data comprises NTHRBL tables of NLNHBL lines of input. Each table must terminate by a slash (/).

Each line contains the items detailed below:

1. Polymer concentration in the solution
   The values should increase monotonically down the column.
   UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scc (LAB)

2. Tortuosity of the porous media, $\delta$.
   UNITS: dimensionless
   DEFAULT: SQRT(2) = $\sqrt{2}$ = 1.414

3. Fluid flow behavior index, or power law exponent, $n$.
   UNITS: dimensionless
   DEFAULT: 1.0

4. Yield stress of the polymer solution, $\tau_0$.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB)
   DEFAULT: 0.0

Example

With NTHRBL=1 and NLNHBL=2:

```
FHERCHBL
   -- Cp.  $\delta$  n  $\tau_0$
   0.0  1.414  1.0  0.0
   10.0  1.414  0.6  0.5 /
```
FIELD

Field units are to be used

This indicates that field units are to be used.

Note: The default unit convention is METRIC.

See "Units" in the ECLIPSE Technical Description for further details.

The keyword has no associated data.

Keywords

790
FIELDSEP

**Introduces a field separator**

This keyword introduces a field separator, used to define separator oil and gas in place. Several lines of FIELDSEP data may be used to build up a multistage separator. The maximum number of stages can be defined by the WELLDIMS keyword. Stages must be specified in increasing order.

If a FIELDSEP keyword is entered, it specifies the default separator conditions for the wells.

The default limit on the number of stages per separator is 5. This may be increased using the 5th item in the WELLDIMS keyword. The default limit on the number of separators (including the FIELDSEP separator) is 4. This may be increased using the 8th item in the WELLDIMS keyword.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated by a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. The stage index.
2. The field separator stage temperature.
   
   **UNITS:** °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
   
   **DEFAULT:** 15.56 °C (METRIC, LAB and PVT-M), 60 °F (FIELD)
3. The field separator stage pressure.
   
   **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
   
   **DEFAULT:** 1.01325 barsa (METRIC), 14.6959 psia (FIELD), 1 atma (LAB and PVT-M)
4. This defines the destination of the liquid output from the field separator.
   
   For all but the last stage in a field separator, this will be to a later stage, usually the next stage.
   
   Setting this item to 0 implies the next stage for all except the last stage, and the stock tank for the last stage.
   
   If -1 is specified for a stage before the last, this volume is added to the stock tank oil. Care must be taken, as this may result in the accumulation of oil volumes at different pressures or temperatures.
   
   **DEFAULT:** 0
5. This defines the destination stage of the vapor output from the field separator.

   This is usually accumulated with the stock tank or field separator vapor, the volume rate being taken by conversion to a volume at standard conditions. This is done when the destination stage is set to 0.

   In the unusual case in which the vapor output from the field separator is fed to a subsequent stage, a non-default value of the destination stage may be entered. Note that, in this case, if intermediate stage separator gas is used as a source of reinjection gas, ECLIPSE 300 will calculate and report export oil rates. This export rate will be less than the produced oil rate which is the ‘formation’ oil rate prior to any reinjection work. The difference then between the formation oil and export oil is the oil vapor that is contained in the reinjected gas that would otherwise have been separated in the subsequent separator stage. See also the discussion under “Export tables” in the ECLIPSE Technical Description.
6. The K-value table number.
This is required if a K-value table rather than an equation of state is being used for the field separator.
A value of 0, indicates no K-value table, and implying an equation of state calculation.
DEFAULT: 0

7. The gas plant table number.
This is required if a gas plant table rather than an equation of state is being used for a stage of the field separator. A value of 0, indicates no gas plant table, and implying an equation of state calculation.
DEFAULT: 0

8. The surface equation of state number.
This should only be specified if multiple equations of state are used. When this option is employed, it may still be defaulted, in which case fluid in place calculation for a cell uses the equation of state for that cell.

If an equation of state index is specified, this forces the use of a given surface equation of state. The value must lie between 1 and the maximum number of equations of state for surface conditions, TABDIMS item 10.

If FIELDSEP defines well separator conditions, a default equation of state index again implies using the value for the cells in which the well is completed.

This is discussed further in "Equations of state regions" in the ECLIPSE Technical Description.

UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

10. Pressure for NGL density evaluation.
UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

If both items 9 and 10 are specified, an equation of state NGL density calculation is performed, rather than using Amagat's law.

Related keywords

For well production, surface volumes are calculated using a well separator, defined by the WSEPCOND keyword. If a well separator is not defined, the group separator defined by the SEPCOND keyword is used. If the group separator is not defined, the field separator defined by this keyword FIELDSEP is used. If the field separator is not defined, a single flash to standard conditions given by STCOND is used.

For fluid-in-place calculations, surface volumes are calculated using region separators, defined by the FIPSEP keyword. If the region separator is not defined, the field separator defined by this keyword FIELDSEP is used. If the field separator is not defined, a single flash to standard conditions given by STCOND is used.
**Examples**

**Example 1**

 Defines a two stage field separator. The first stage has a pressure of 815 psi and a temperature of 80°F. The second stage is connected to the liquid output of the first, and represents stock tank conditions.

```
FIELDSEP
  1  80 815 /
  2  60 14.7 /
```

**Example 2**

 For a much more complex case, the following field separator configuration, in which the first stage uses an equation of state flash, the vapor output feeding a gas plant stage to extract NGLs, and the liquid output feeding a stock tank flash stage.

```
FIELDSEP
  1 60 15   3  2  1* 1* / --Flash first stage, liquid to 3, vapor to 2
  2 15 1.03 -1 1* 1* 1  / --Gas plant second stage for NGL recovery
  3 15 1.03 1* 1* 1* 1* / --Stock tank for oil recovery
```

*Figure 3.5. A complex field separator configuration*
Confirm units of a data set

The keyword should be followed by an item, which specifies the units being used. The data should be terminated by a slash (/).

The possible values of the option are:

- **METRIC**   The data is specified using the same unit system as for the METRIC keyword.
- **FIELD**    The data is specified using the same unit system as for the FIELD keyword.
- **LAB**      The data is specified using the same unit system as for the LAB keyword.
- **PVT-M**    This option is for ECLIPSE 300 only. The data is specified using the same unit system as for the PVT-M keyword.

There is no default value.

*Note:* This keyword does **not** provide conversion between different unit systems. It is intended to facilitate checking where, for instance, a data set may contain several sections in separate files which were created by different programs possibly using different units. All occurrences of FILEUNIT in a data set have to match.

**Example**

Specify that the data is in FIELD units.

```
FILEUNIT
FIELD /
```
FILLEPS

Requests that saturation end points for all grid blocks are written to the INIT file

This keyword affects the output of grid block end point data in the INIT file. Normally, the values of SWL, SWCR, SOWCR and similar keywords are written to the INIT file only when end point scaling is on (keyword ENDSCALE in the RUNSPEC section) or if (ECLIPSE 100) the initial mobile fluid correction is on (switch 'MOBILE' set in the RUNSPEC keyword EQLOPTS). The grid block endpoints are also by default only output for those cells for which the user has explicitly defined the rescaled end points with the appropriate keywords in the PROPS section (including keywords SWL, SWCR, SOWCR). Grid blocks for which rescaling data has not been explicitly defined are indicated as undefined, meaning that the original saturation tables (for example, the families of keywords including SWOF/SGOF, or SWFN/SGFN/SOF3) are to be employed for these blocks without rescaling.

The keyword FILLEPS instructs ECLIPSE to output the actual end points for each block, whether taken from the saturation tables or rescaled values, whether or not the end point scaling option is used. FILLEPS has no effect if the INIT keyword is not present in the GRID section.

The keyword has no associated data.

Example

```
FILLEPS
```
**FIP**

### Additional sets of fluid-in-place region numbers

This keyword is used to specify an array of region numbers for extra sets of fluid-in-place regions, in addition to the standard set defined with keyword **FIPNUM**.

The keyword name should consist of two segments:

1. Characters 1-3 must be the character string **FIP**
2. Characters 4-8 must be the name of the set of fluid-in-place regions. Note that 5 characters may be used, but only the first 3 characters are significant; hence the first 3 characters should be unique.

The keyword should be followed by one integer for every grid block in the current input box, specifying the fluid-in-place region number to which it belongs in this set. The region numbers should not be less than 1 or greater than **NTFIP** (see keyword **REGDIMS**). The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

The total number of FIP type keywords in a run (including **FIPNUM**) must not exceed **NMFIPR** (set by keyword **REGDIMS**).

The FIP type keywords can also be used in conjunction with the **COPY**, **ADD** and **EQUALS** keywords.

Any grid blocks not assigned a region number in each set of FIP regions when the end of the **REGIONS** section is reached are assigned to FIP region 1.

A set of balance sheets is produced for each set of fluid-in-place regions supplied using the FIP keyword, when the FIP mnemonic is set >2 in **RPTSCHED**. The first set of regions in the report is the set constructed with the **FIPNUM** keyword. The subsequent balance sheets report the additional fluid-in-place regions constructed with the FIP keyword. These have a title mnemonic of the form **BALnnnnn**, where nnnn is the name of the set of fluid-in-place regions specified with the keyword **FIPnnnnn**.

**SUMMARY** section quantities can be written to the summary files by using one of the summary keywords for region output (keywords beginning with R) compounded with the first 3 characters of the name of the set of fluid-in-place regions. Summary keywords having less than 5 characters should be extended to 5 characters with underscores before adding the 3-character region set name; for example,

```
ROIPLFST for the oil-in-place in the liquid phase for region 2 of 2 / the set of regions named FST.
ROIP_SND for the total oil-in-place in all the regions in the / set named SND.
```

**ECLIPSE 100 only**

See also keyword **FIPOWG**, which automatically constructs a set of FIP regions to represent the gas, oil and water zones at equilibration.

### Example

With **NMFIPR ≥ 3** and **NTFIP=4** specified in keyword **REGDIMS**; **NDIVIX=8, NDIVIY=6** and **NDIVIZ=3** specified in **RUNSPEC** keyword **DIMENS**; and no input **BOX** set:

```
-- FIPNUM by layer, for standard fluid-in-place region
FIPNUM
48*1
48*2
48*3 /
-- FIP by area, X strips (region name XAREA)
FIPXAREA
```
-- FIP by area, Y strips (region name YAREA)
FIPYAREA
8*1 16*2 16*3 8*4
8*1 16*2 16*3 8*4 /
### Fluid-in-place region numbers

The keyword should be followed by one integer for every grid block in the current input box, specifying the fluid-in-place region to which it belongs.

A balance sheet showing fluid in place and cumulative flows to and from wells and other regions is produced for each fluid-in-place region at every report time, if the FIP mnemonic in the RPTSCHED keyword is set >1.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

Any grid blocks not assigned a value of FIPNUM when the end of the REGIONS section is reached will be assigned to region 1.

The region numbers should not be less than 1 or greater than NTFIP (see keyword REGDIMS or TABDIMS). The data must be terminated by a slash (/).

This keyword defines the standard set of fluid-in-place regions. Additional sets of fluid-in-place regions can be defined, if required, with the FIP keyword. Each set of fluid-in-place regions produces its own set of balance sheets, independently of the other sets of fluid-in-place regions. The number of sets of fluid-in-place regions must not exceed NMFIPR (keyword REGDIMS).

**Note:** The pore volume reported in the balance sheet will include the pore volume of any aquifers associated with the region. Aquifer cells with no explicit region number are assigned to region 1.

See also the keyword FIPOWG.

### Example

With NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS, and no input BOX set:

```plaintext
  FIPNUM
  40*2 8*1
  40*3 8*1
  40*4 8*1
  40*5 8*1
  40*6 8*1 /
```

Keywords

798
Sets up a family of FIP regions representing the original GAS, OIL and WATER zones

This keyword requests that a family of fluid-in-place regions should be set up automatically to represent the original gas, oil and water zones at equilibration.

The keyword has no associated data.

The total number of FIP type keywords in a run (including FIPNUM) must not exceed NMFIPR, and NTFIP should be set to at least a value of 3 because there will be three regions in this family. (Both NMFIPR and NTFI are set by keyword REGDIMS.)

Three regions are set up according to the depth of the cells.

Region:
1. Gas zone, containing all cells with depths above the GOC specified in item 5 of the EQUIL keyword.
2. Oil zone, containing the cells with depths between the GOC and WOC.
3. Water zone, containing all cells with depths below the WOC specified in item 3 of the EQUIL keyword.

This family of regions is treated in a similar way to any other FIP region family constructed manually with the FIP keyword. A set of balance sheets is produced for the FIPOWG fluid-in-place regions, when the FIP mnemonic is set >2 in RPTSCHED. These have a title mnemonic BALOWG.

SUMMARY section quantities can be written to the Summary files by using one of the Summary keywords for region output (keywords beginning with R) compounded with the 3 characters OWG. SUMMARY keywords having less than 5 characters should be extended to 5 characters with underscores before adding the 3-character region set name; for example,

| ROIPLOWG | for the oil-in-place in the liquid phase for the original oil zone. |
| ROIP_OWG | for the total oil-in-place in all three regions |

See also the FIPNUM and FIP keywords.

Example

| FIPOWG |  |
**FIPSEP**

**Introduces a fluid-in-place region separator**

This keyword introduces a separator that will be used to define separator oil- and gas-in-place volumes for a specific fluid-in-place region. Several lines of FIPSEP data may be used to build up a multi-stage separator. The maximum number of stages can be defined by the **WELLDIMS** keyword. Stages must be specified in increasing order.

The default limit on the number of stages per separator is 5. This may be increased using the item 5 in the **WELLDIMS** keyword. The default limit on the number of separators (including the **FIELDSEP** separator) is 4. This may be increased using the 8th item in the **WELLDIMS** keyword.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated by a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. The number of the fluid-in-place region in which this separator is used for oil-and gas-in-place volume calculation. The region numbers should not be less than 1 or greater than **NTFIP** (see keyword **REGDIMS**).

2. The stage index.

3. The separator stage temperature.
   
   **UNITS:** °C (**METRIC**), °F (**FIELD**), °C (**LAB**), °C (**PVT-M**)  
   **DEFAULT:** 15.56 °C (**METRIC**, **LAB** and **PVT-M**), 60 °F (**FIELD**)

4. The separator stage pressure.
   
   **UNITS:** barsa (**METRIC**), psia (**FIELD**), atma (**LAB**), atma (**PVT-M**)  
   **DEFAULT:** 1.01325 barsa (**METRIC**), 14.6959 psia (**FIELD**), 1 atma (**LAB** and **PVT-M**)

5. This defines the destination of the liquid output from the separator.
   
   For all but the last stage in a separator, this is to a later stage, usually the next stage.
   
   Setting this item to 0 implies the next stage for all except the last stage, and the stock tank for the last stage.
   
   If -1 is specified for a stage before the last, this volume is added to the stock tank oil. Care must be taken, as this may result in the accumulation of oil volumes at different pressures or temperatures.
   
   **DEFAULT:** 0

6. This defines the destination stage of the vapor output from the separator.

   This is usually accumulated with the stock tank or separator vapor, the volume rate being taken by conversion to a volume at standard conditions. This is done when the destination stage is set to 0.

   In the unusual case in which the vapor output from the separator is fed to a subsequent stage, a non-default value of the destination stage may be entered.

   **DEFAULT:** 0

7. The K-value table number.
This is required if a K-value table rather than an equation of state is being used for the separator. A value of 0 indicates no K-value table, and implies an equation of state calculation.

DEFAULT: 0

8. The gas plant table number.

This is required if a gas plant table rather than an equation of state is being used for a stage of the separator. A value of 0 indicates no gas plant table, and implies an equation of state calculation.

DEFAULT: 0

9. The surface equation of state number.

This should only be specified if multiple equations of state are used. When this option is employed, it may still be defaulted, in which case the fluid in place calculation for a cell uses the equation of state for that cell.

If an equation of state index is specified, this forces the use of a given surface equation of state. The value must lie between 1 and the maximum number of equations of state for surface conditions, TABDIMS item 10.

This is discussed further in “Equations of state regions” in the ECLIPSE Technical Description.

10. Temperature for natural gas liquid (NGL) density evaluation.

UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

11. Pressure for NGL density evaluation.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

If both items 10 and 11 are specified, an equation of state NGL density calculation is performed, rather than using Amagat's law.

Related keywords

For fluid-in-place calculations, surface volumes are calculated using region separators, defined by this keyword FIPSEP. If the region separator is not defined, the field separator FIELDSEP is used. If the field separator is not defined, a single flash to standard conditions given by STCOND is used.

Notes

- This keyword allows different fluid-in-place regions to use different separators for calculating the in-place volumes of oil and gas, as reported in the print file. If this keyword is omitted, all regions use the FIELDSEP separator for this calculation.

- The in-place total volumes for the whole field are calculated using the FIELDSEP separator, and so these oil and gas totals do not in general equal the sum of the oil and gas volumes from fluid-in-place regions with differing separators.

- The presence of this keyword does not alter the default separator assignment for wells; this is still taken as the FIELDSEP separator, regardless of which fluid-in-place region the well is in.

Example

Defines a two-stage separator for the fluid-in-place region number 1, a one-stage separator for region number 2, and a three-stage separator for region number 3.
ECLIPSE Reference Manual

Keywords

802
**FLUXNUM**

Identifies extent of each flux region

The FLUXNUM keyword has two uses. Firstly, it provides a convenient way of defining a region in the GRID section for subsequent modification of its data using the EQUALREG, ADDREG, COPYREG, MULTIREG, MULTREGP and MULTREGT keywords. Secondly, the keyword is used with the flux boundary option to define regions that can be run as separate models with boundary fluxes defined in a previous full field run.

The keyword should be followed by one integer for each grid cell in the full field model, specifying the flux region number to which it belongs. The region numbers should not exceed the value of NTFREG entered using keyword REGDIMS in the RUNSPEC section (item 4). The data must be terminated by a slash (/).

The FLUXNUM array can be copied into the REGIONS section arrays (for example FIPNUM and SATNUM) using the COPY keyword. It is recommended that NTFREG should be set greater than 1 if this facility is to be used. If a zero FLUXNUM region is copied in to a REGIONS array then the destination array will be set to 1.

**ECLIPSE 100**

Any cells not explicitly assigned a FLUXNUM value will be defaulted to a value of 1.

**ECLIPSE 300**

Any cells not explicitly assigned a FLUXNUM value will be defaulted to a value of -1.

Set FLUXNUM to zero explicitly for any cells not required in flux regions.

The FLUXNUM keyword should be used before the EQUALREG, ADDREG, COPYREG, MULTIREG and MULTREGT keywords are used to modify grid properties within a flux region.

**ECLIPSE 100 only**

The FLUXNUM keyword cannot be used inside a CARFIN/RADFIN ... ENDFIN bracket.

**ECLIPSE 100**

For use with the dual porosity option, only the matrix cells or the fracture cells need to be input. The cell values in the other porosity will be copied automatically. If the user wants to specify matrix cells and fracture cells separately, then item 117 in OPTIONS should be set greater than 1. This will also affect use of the MULTNUM, PINCHNUM or OPERNUM keywords.

**ECLIPSE 300**

For use with dual porosity option, both matrix and fracture cells should be provided. FLUXNUM use with MULTREGT or MULTREGP, will automatically copy all matrix cells to the corresponding fracture cells, so dummy fracture values could be input. For other FLUXNUM uses, there is no copying of data from the matrix to fracture cells.

**Flux option cases**

See "Flux Boundary Conditions" in the ECLIPSE Technical Description for more information on this option.

The FLUXNUM keyword must be used in both the full-field (DUMPFLUX) run and the reduced (USEFLUX) run to identify the cells belonging to each flux region. The DUMPFLUX or USEFLUX keyword must have been entered previously.

For runs in which the full field has just a single flux region (NTFREG=1), the cells inside the flux region should have a region number of 1, and 0 for the cells outside. For runs involving multiple flux regions (NTFREG>1), the region numbers should lie between 1 and NTFREG, and the FLUXREG keyword is used in the reduced run to indicate which flux regions are active.

**ECLIPSE 100 only**

(A negative region number will automatically be converted to positive.)

The contents of the FLUXNUM keyword in a reduced run must be identical with the original full-field run.
Examples

Example 1

To establish a single 2*2*2 flux region in a 4*4*3 field:

```
FLUXNUM
1 1 0 0
1 1 0 0
0 0 0 0
0 0 0 0
1 1 0 0
1 1 0 0
0 0 0 0
0 0 0 0
16*0 /
```

Example 2

To define 9 separate flux regions in a 4*4*3 field:

```
FLUXNUM
1 1 2 2
1 1 2 2
3 3 3 3
4 4 4 4
5 5 5 5
6 7 8 8
6 7 8 8
6 7 8 8
16*9 /
```
Options for flux boundary runs

This keyword should be used in the full-field run to tell ECLIPSE whether the reduced run will be a thermal run or not.

The FLUXOPTS keyword should be followed by one of the following values:

YES  The simulator will output the necessary information to run a thermal reduced run

NO   The simulator will not output the necessary information to run a thermal reduced run

DEFAULT: NO

The data should be terminated by a slash (/).

Note: There is no need to use this keyword if the base run is thermal or there is no intention of running a thermal sector model from the base run.

Example

The reduced run will be a thermal run

```
FLUXOPTS
YES /
```
Identifies which flux regions are active

The FLUXREG keyword should be used in a reduced (USEFLUX) run where there is more than one flux region in the FLUXNUM keyword. It tells the simulator which flux regions are active in the reduced run. The keyword should be entered after the USEFLUX keyword.

The keyword should be followed by up to NTFREG (see item 4 in the RUNSPEC section keyword REGDIMS) integers, terminated with a slash (/). The integers form a list of the flux regions that are active. The remaining flux regions that are not on the list are made inactive, and their flows into the active regions are read from the Flux file.

The keyword is not required in full-field (DUMPFLUX) runs, or in runs where there is only one flux region in the full field (NTFREG = 1).

See also "Flux boundary conditions" in the ECLIPSE Technical Description.

Example

To perform a reduced run on a sector of the field defined by flux regions 2, 3 and 5:

```
FLUXREG
  2 3 5 /
```
**Specify type of flux boundary condition**

The `FLUXTYPE` keyword specifies the type of boundary conditions to be used for the reduced run.

The boundary conditions supplied from the full field run are used in the reduced run. ECLIPSE allows two types of boundary condition:

- Flow boundary conditions supply the boundary fluxes for each phase from the full field run.
- Pressure boundary conditions use the pressures, saturations, Rs and Rv values from the full field run as boundary conditions for the reduced run in ‘halo’ cells surrounding the reduced region activated by `USEFLUX` and `FLUXREG`.

The same type of boundary condition must be used in both the full field run (`DUMPFLUX`) and the reduced run (`USEFLUX`).

The keyword is followed by a single item to select the chosen boundary condition, terminated by a slash (/).

1. **Type of boundary condition**
   - `'FLUX'` Use flows of oil, water and gas from the full field run as boundary conditions on the reduced run
   - `'PRESSURE'` Use pressures and saturations from the full field run in ‘halo’ cells adjoining the reduced flux region as boundary conditions on the reduced run.

   **DEFAULT:** `'FLUX'`

   If this keyword is not present in the data file, FLUX boundary conditions are used.

**Example**

To establish pressure boundary conditions:

```
   FLUXTYPE
       'PRESSURE' /
```

---

**Keywords**

807
**FMTHMD**

**Indicates that HMD file is to be formatted**

Declares that the HMD output file, used by the ECLIPSE Gradient option, containing the derivatives is to be formatted.

The default is unformatted.

See "Gradient option" in the *ECLIPSE Technical Description* for further information.

The keyword has no associated data.

This option is for ECLIPSE 100 only. If an RFT gradients file `.GRFT` is produced, this is also formatted.

**Keywords**

- ECLIPSE 100
- ECLIPSE 300
- SPECIAL
- RUNSPEC
- GRID
- EDIT
- PROPS
- REGIONS
- SOLUTION
- SUMMARY
- SCHEDULE
FMTIN Indicates that input files are formatted

This indicates that input files which may be either formatted or unformatted, such as restart files, are to be formatted. The default, if this keyword is not specified, is to use unformatted files, as this reduces file size and processing time.

FMTIN contains no items.
FMTOUT

Indicates that output files are formatted

This indicates that output files which may be either formatted or unformatted are to be formatted. This includes the SMSPEC, SUMMARY, GRID, INIT and RESTART files (see "File handling in ECLIPSE" in the ECLIPSE Technical Description).

The default is unformatted, as this reduces file size and processing time.

FMTOUT contains no items.

The format of ECLIPSE 300 SAVE files is controlled by a separate keyword, FMTSAVE.

<table>
<thead>
<tr>
<th>Keywords</th>
</tr>
</thead>
<tbody>
<tr>
<td>ECLIPSE 100</td>
</tr>
<tr>
<td>ECLIPSE 300</td>
</tr>
<tr>
<td>SPECIAL</td>
</tr>
<tr>
<td>RUNSPEC</td>
</tr>
<tr>
<td>GRID</td>
</tr>
<tr>
<td>EDIT</td>
</tr>
<tr>
<td>PROPS</td>
</tr>
<tr>
<td>REGIONS</td>
</tr>
<tr>
<td>SOLUTION</td>
</tr>
<tr>
<td>SUMMARY</td>
</tr>
<tr>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>

Keywords: 810
**FMTSAVE**  Indicates that save files are formatted

This indicates that ECLIPSE 300 *SAVE* files are to be formatted. By default these files are unformatted.

FMTSAVE contains no items.

<table>
<thead>
<tr>
<th>ECLIPSE 100</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>ECLIPSE 300</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SPECIAL</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>RUNSPEC</td>
<td></td>
</tr>
<tr>
<td></td>
<td>GRID</td>
<td></td>
</tr>
<tr>
<td></td>
<td>EDIT</td>
<td></td>
</tr>
<tr>
<td></td>
<td>PROPS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>REGIONS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SOLUTION</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SUMMARY</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SCHEDULE</td>
<td></td>
</tr>
</tbody>
</table>
Enable FOAM option

This indicates that the ECLIPSE 100 foam model is required to allow the modeling of foam injection processes.

See "Foam model" in the ECLIPSE Technical Description for further information.

The keyword has no associated data.

<table>
<thead>
<tr>
<th>x</th>
<th>ECLIPSE 100</th>
<th>ECLIPSE 300</th>
<th>SPECIAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>RUNSPEC</td>
<td>GRID</td>
<td>EDIT</td>
</tr>
<tr>
<td></td>
<td>PROPS</td>
<td>REGIONS</td>
<td>SOLUTION</td>
</tr>
<tr>
<td></td>
<td>SUMMARY</td>
<td>SCHEDULE</td>
<td></td>
</tr>
</tbody>
</table>
**FOAMADS**

**Foam adsorption functions**

The data comprises NTSFUN (see keyword **TABDIMS**) tables of foam adsorption functions, each terminated by a slash (/), describing the adsorption of foam by the rock formation. See "Foam model" in the **ECLIPSE Technical Description** for further information.

Each table consists of 2 columns of data:

1. The local foam concentration in the solution surrounding the rock. The values should increase monotonically down the column.
   
   If item 1 of **FOAMOPTS** is set to ‘GAS’:
   
   UNITS: kg/sm³ (METRIC), lb/Mscf (FIELD), gm/scc (LAB)
   
   If item 1 of **FOAMOPTS** is set to ‘WATER’:
   
   UNITS: kg/sm³ (METRIC), lb/stb (FIELD), gm/scc (LAB)

2. The corresponding saturated concentration of foam adsorbed by the rock formation. These values should be level, or increase down the column.

   UNITS: kg/kg (METRIC), lb/lb (FIELD), gm/gm (LAB)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see keyword **TABDIMS**).

**CAUTION:** The first row in the adsorption tables should consist of a pair of zeroes.

See also **FOAMROCK**, **SURFADS**, **PLYADS**, **PLYADSS** and **ADSORP**.

**Example**

With NTSFUN=2 and SSFUN≥6)

```
   FOAMADS
    .0000  .00000
    .0001  .00001
    .0003  .00005
    .0005  .00007
    .0008  .00008
    .0010  .00008
    /        
    .0000  .00000
    .0010  .00010
    /        
```

---

**Keywords**

---
### FOAMDCYO

**Foam decay data as a function of oil saturation**

The FOAMDCYO keyword inputs data describing the decay of foam as a function of oil saturation. See "Foam model" in the ECLIPSE Technical Description for further information.

The data comprises NTSFUN (see keyword TABDIMS) tables of foam decay data, each terminated by a slash (/). Each table consists of 2 columns of data:

1. The local oil saturation.
   - The values should increase monotonically down the column.
2. The corresponding decay half-life.
   - UNITS: day (METRIC), day (FIELD), hr (LAB)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see keyword TABDIMS).

Note that if the FOAMDCYW keyword is also present, the decay half-life is taken to be the minimum of the two values for each cell.

See also keywords FOAMDCYW and FOAMADS.

### Example

With NTSFUN=2 and NSSFUN ≥ 3:

```plaintext
FOAMDCYO
   -- Constant half-life
   --
   0.0  12.0
   1.0  12.0  /
   --
   -- Reduced half-life if oil is present.
   --
   0.0  15.0
   0.1  5.0
   1.0  5.0  /
```

---

**Keywords**

814
FOAMDCYW  Foam decay data as a function of water saturation

The FOAMDCYW keyword inputs data describing the decay of foam as a function of water saturation. See “Foam model” in the ECLIPSE Technical Description for further information.

The data comprises NTSFUN (see keyword TABDIMS) tables of foam decay data, each terminated by a slash (/). Each table consists of 2 columns of data:

1. The local water saturation.
   The values should increase monotonically down the column.

2. The corresponding decay half-life.
   UNITS: day (METRIC), day (FIELD), hr (LAB)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see keyword TABDIMS).

Note that if the FOAMDCYO keyword is also present, the decay half-life is taken to be the minimum of the two values for each cell.

See also keywords FOAMDCYO and FOAMADS.

Example

With NTSFUN=2 and NSSFUN ≥ 2:

```
FOAMDCYW
-- Constant half-life
--
  0.0  15.0
  1.0  15.0 /
--
-- Reduced half-life if water is present.
--
  0.0  15.0
  1.0   5.0 /
```
FOAMFCN  Gas mobility reduction dependence upon capillary number

The data comprises NTSFUN (see keyword TABDIMS) tables defining the parameters which control the gas mobility reduction due to foam as a function of capillary number. Each table consists of one row of data containing two items which should be terminated with a slash (/):

1. The reference capillary number.
2. An exponent controlling the steepness in the change of mobility according to the ratio of reference to calculated capillary numbers.
   
   DEFAULT: 1.0

The first item in the first table must be specified. The second item may be defaulted. If the second or subsequent tables are completely defaulted, the associated parameters will be copied from the previous table.

This keyword is only relevant if the foam model is used with the functional gas mobility reduction model.

In ECLIPSE 100 this requires the FOAM option to be specified in the RUNSPEC section and FUNC to be set for item 2 of the FOAMOPTS keyword.

In ECLIPSE 300 the foam model is activated by using an additional water component COMPW to represent the foam surfactant CWTYPE. ECLIPSE 300 only supports the functional form of the gas mobility model, so this need not be specified.

In both simulators, the FOAMFSC keyword must be specified with the functional gas mobility reduction model. See "Foam model" in the ECLIPSE Technical Description for further information.

For the functional model of gas mobility reduction, the mobility reduction factor $M_{rf}$ is expressed in the form:

$$M_{rf} = \frac{1}{1 + (M_r \cdot F_s \cdot F_w \cdot F_o \cdot F_c)}$$

where

$M_r$ is the reference mobility reduction factor (FOAMFRM keyword).

$F_s$ is the mobility reduction factor dependence upon surfactant concentration (FOAMFSC keyword).

$F_w$ is the mobility reduction factor dependence upon water saturation (FOAMFSW keyword).

$F_o$ is the mobility reduction factor dependence upon oil saturation (FOAMFSO keyword).

$F_c$ is the mobility reduction factor dependence upon capillary number (this keyword).

The mobility reduction factor functional dependence upon capillary number $F_c$ is expressed in the form:

$$F_c = \left(\frac{N_c^r}{N_c}\right)^e$$  \hspace{1cm} Eq. 3.72

where
\( N_c \) is the capillary number.

\( N_c' \) is the reference capillary number.

\( \epsilon_c \) is an exponent which controls the steepness of the transition about the point where \( N_c = N_c' \).

The capillary number is a dimensionless parameter which provides a measure of the ratio of viscous forces to capillary forces. The calculation of the capillary number is described in "Capillary number" in the ECLIPSE Technical Description. The viscous forces are proportional to the velocity, which is calculated from the flows into and out of the cell, while the capillary forces are proportional to the gas-water surface tension defined by the FOAMFST keyword.

Several simplifications are made when calculating the velocity term:

**ECLIPSE 100**
- In ECLIPSE 100 non-neighbor flows are not included

**ECLIPSE 300**
- In ECLIPSE 300 non-neighbor flows are included, except
  - user-defined non-neighbor flows (NNC keyword)
  - non-neighbor flows stored in flux boundary files
  - non-neighbor flows arising from grid coarsening
- Well flows are not included.

The gas-water interfacial tension is tabulated as function of the foam concentration using the FOAMFST keyword. The contribution of the capillary number mobility reduction component to the overall gas mobility reduction is determined by the ratio of reference to calculated grid block capillary number and the value of the exponent which controls the steepness of the change about the transition point \( N_c = N_c' \). This keyword is optional; however, if this keyword is specified the FOAMFST keyword must also be specified. If this keyword is not specified the component of the overall mobility reduction factor which models the functional dependence upon capillary number will default to 1.0. See also the FOAMFRM, FOAMFSC and FOAMFSW keywords.

The capillary number functional mobility reduction term is only applied in grid blocks which contain foam with a concentration which is greater than or equal to the minimum concentration specified by item 3 of the FOAMFSC keyword.

**Example**

With \( \text{NTSFUN}=5 \)

```plaintext
FOAMFCN
  1.0E-07 /
  2.0E-07 1.5 /
  / 1.0E-07 3.0/
```

Keywords

817
**FOAMFRM  Gas mobility reduction dependence upon reference mobility reduction**

The data comprises NTSFUN (see keyword **TABDIMS**) tables defining the parameters which control the gas mobility reduction as a function of the reference mobility reduction factor. Each table consists of one row of data containing one item which should be terminated with a slash(/):

1. The reference mobility reduction factor. This value must be greater than zero and should typically lie in the region of 1 to 100

The single item in the first table must be specified. If the single item is omitted from the second or subsequent tables, the reference mobility reduction factor will be copied from the previous table.

This keyword is only relevant if the foam model is used with the functional gas mobility reduction model.

In ECLIPSE 100 this requires the **FOAM** option to be specified in the RUNSPEC section and **FUNC** to be set for item 2 of the **FOAMOPTS** keyword.

In ECLIPSE 300 the foam model is activated by using an additional water component **COMPW** to represent the foam surfactant **CWTYPE**. ECLIPSE 300 only supports the functional form of the gas mobility model, so this need not be specified.

In both simulators, the **FOAMFSC** keyword must be specified with the functional gas mobility reduction model. See “Foam model” in the ECLIPSE Technical Description for further information.

For the functional model of gas mobility reduction, the mobility reduction factor $M_{rf}$ is expressed in the form:

$$M_{rf} = \frac{1}{1 + (M_r \cdot F_s \cdot F_w \cdot F_o \cdot F_c)}$$

where

- $M_r$ is the reference mobility reduction factor (this keyword).
- $F_s$ is the mobility reduction factor dependence upon surfactant concentration (**FOAMFSC** keyword).
- $F_w$ is the mobility reduction factor dependence upon water saturation (**FOAMFSW** keyword).
- $F_o$ is the mobility reduction factor dependence upon oil saturation (**FOAMFSO** keyword).
- $F_c$ is the mobility reduction factor dependence upon capillary number (**FOAMFCN** keyword).

This keyword is optional. If this keyword is not specified the reference mobility reduction factor will default to 1.0. See also the **FOAMFCN**, **FOAMFSC**, **FOAMFSO** and **FOAMFSW** keywords. The reference mobility reduction factor is only applied in grid blocks which contain foam with a concentration which is greater than or equal to the minimum concentration specified by item 3 of the **FOAMFSC** keyword.

**Example**

With NTSFUN=5

```
FOAMFRM
50 /
```

**Keywords**
The data comprises NTSFUN (see keyword TABDIMS) tables defining the parameters which control the gas mobility reduction as a function of foam surfactant concentration. Each table consists of one row of data and should be terminated with a slash(/):

1. The reference foam surfactant concentration above which a strong foam can form.
   
   **UNITS:** kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/cc (LAB)

2. An exponent controlling the steepness in the change of mobility reduction due to surfactant concentration.
   
   **DEFAULT:** 1.0

3. The minimum surfactant concentration for which gas mobility reduction due to the presence of foam will be calculated.
   
   **UNITS:** kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/cc (LAB)
   
   **DEFAULT:** $10^{-20}$

4. The minimum water saturation below which foam has no effect.
   
   **DEFAULT:** $10^{-6}$

The first item in the first table must be specified. The second and third items may be defaulted.

The fourth term is only used in ECLIPSE 300. It is intended to prevent small numerical errors from causing convergence problems: when the water saturation is very low, a small amount of foam surfactant will lead to a large surfactant concentration, which could have a spuriously large effect on the gas mobility.

If the second or subsequent tables are completely defaulted, the associated parameters will be copied from the previous table.

This keyword is only relevant if the foam model is used with the functional gas mobility reduction model.

In ECLIPSE 100 this requires the FOAM option to be specified in the RUNSPEC section and FUNC to be set for item 2 of the FOAMOPTS keyword.

In ECLIPSE 300 the foam model is activated by using an additional water component COMPW to represent the foam surfactant. The CWTYPE keyword is used to identify which water component(s) acts as the foam surfactant (SURFF). ECLIPSE 300 only supports the functional form of the gas mobility model, so this need not be specified.

See "Foam model" in the ECLIPSE Technical Description for further information.

For the functional model of gas mobility reduction, the mobility reduction factor $M_{rf}$ is expressed in the form:

$$M_{rf} = \frac{1}{1 + (M_r \cdot F_s \cdot F_w \cdot F_o \cdot F_c)}$$

where

- $M_r$ is the reference mobility reduction factor (FOAMFRM keyword).
$F_s$ is the mobility reduction factor dependence upon surfactant concentration (this keyword).

$F_w$ is the mobility reduction factor dependence upon water saturation (FOAMFSW keyword).

$F_o$ is the mobility reduction factor dependence upon oil saturation (FOAMFSO keyword).

$F_c$ is the mobility reduction factor dependence upon capillary number (FOAMFCN keyword).

The mobility reduction factor dependence upon surfactant concentration $F_s$ is expressed in the form:

$$F_s = \left( \frac{C_s}{C_s^r} \right)^{\epsilon_s}$$

where

$C_s$ is the surfactant concentration.

$C_s^r$ is the reference surfactant concentration such that $C_s < C_s^r$ denotes a weak foam and $C_s > C_s^r$ denotes a strong foam.

$\epsilon_s$ is an exponent which controls the steepness of the transition about the point where $C_s = C_s^r$.

The contribution of this component to the overall gas mobility reduction is determined by the reference surfactant concentration, above which the presence of surfactant becomes significant in the creation of foam, and the value of the exponent which controls how quickly this factor changes about the transition point $C_s = C_s^r$. In addition, gas mobility reduction due to the presence of foam will only be calculated for grid blocks in which the foam concentration is greater than or equal to the specified minimum value.

It is important to note that if $\epsilon_s < 1$, the slope of $F_s$ will be infinite at $C_s = 0$. In this case, small surfactant concentrations may have a significant effect on the mobility, particularly if the reference concentration $C_s^r$ is also small. Therefore it may be necessary to use item 3 of this keyword to set a minimum surfactant concentration to prevent “noise” in the form of small numerical errors from affecting the simulation.

This keyword is mandatory if the functional model of gas mobility reduction due to foam has been specified. See also the FOAMFCN, FOAMFRM, FOAMFSO and FOAMFSW keywords.

**Example**

With NTSFUN = 5 specified in keyword TABDIMS:

```
FOAMFSC
  0.001 / 0.002 1.01 / / 0.001 0.9 / 0.002 1.02 /
```

**Keywords**

821
Gas mobility reduction dependence upon oil saturation

The data comprises NTSFUN (see keyword TABDIMS) tables defining the parameters which control the gas mobility reduction due to foam as a function of oil saturation. Each table consists of one row of data containing two items which should be terminated with a slash(/):

1. The maximum oil saturation above which foam ceases to be effective.
2. An exponent controlling the steepness in the change of mobility reduction due to oil saturation.
   DEFAULT: 1.0

The first item in the first table must be specified. The second item may be defaulted. If the second or subsequent tables are completely defaulted, the associated parameters will be copied from the previous table.

This keyword is only relevant if the foam model is used with the functional gas mobility reduction model.

In ECLIPSE 100 this requires the FOAM option to be specified in the RUNSPEC section and FUNC to be set for item 2 of the FOAMOPTS keyword.

In ECLIPSE 300 the foam model is activated by using an additional water component COMPW to represent the foam surfactant CWTYPE. ECLIPSE 300 only supports the functional form of the gas mobility model, so this need not be specified.

In both simulators, the FOAMFSC keyword must be specified with the functional gas mobility reduction model. See "Foam model" in the ECLIPSE Technical Description for further information.

For the functional model of gas mobility reduction, the mobility reduction factor $M_f$ is expressed in the form:

$$M_f = \frac{1}{1 + (M_r \cdot F_s \cdot F_w \cdot F_o \cdot F_c)}$$

where

$M_r$ is the reference mobility reduction factor (FOAMFRM keyword).

$F_s$ is the mobility reduction factor dependence upon surfactant concentration (FOAMFSC keyword).

$F_w$ is the mobility reduction factor dependence upon water saturation (FOAMFSW keyword)

$F_o$ is the mobility reduction factor dependence upon oil saturation (this keyword).

$F_c$ is the mobility reduction factor dependence upon capillary number (FOAMFCN keyword)

The mobility reduction factor functional dependence upon oil saturation $F_o$ is expressed in the form:

$$F_o = \left( \frac{S_o^m - S_o}{S_o^m} \right)^{e_o}$$

where
$S_o$ is the oil saturation.

$S_o^m$ is the maximum oil saturation above which the foam ceases to be effective.

$e_o$ is an exponent which controls the steepness of the transition about the point where $S_o = S_o^m$.

The contribution of this component to the overall gas mobility reduction is determined by the maximum oil saturation, above which the foam ceases to be effective, and the value of the exponent which controls the steepness of the change about the transition point $S_o = S_o^m$. This keyword is optional. If this keyword is not specified the component of the overall mobility reduction factor which models the functional dependence upon oil saturation will default to 1.0. See also the FOAMFCN, FOAMFSC, FOAMFRM and FOAMFSW keywords.

The oil saturation functional mobility reduction term is only applied in grid blocks which contain foam with a concentration which is greater than or equal to the minimum concentration specified by item 3 of the FOAMFSC keyword.

**Example**

With NTSFUN =5

```
FOAMFSO
   -- F_o = ((S_o^m - S_o) / S_o^m)**e_o
   -- S_o^m  e_o
   0.4     1.      /
   0.4     1.01    /
   / 0.3     0.9     /
   / 0.3     1.02    /
```

**Keywords**
FOAMFST  Gas-water surface tension versus foam surfactant concentration

This keyword supplies tables of gas-water surface tension as a function of foam surfactant concentration in the water.

The data comprises NTSFUN (see keyword TABDIMS) tables of gas-water surface tension versus foam surfactant concentration. Each table consists of two columns of data, terminated with a slash (/).

1. The foam surfactant concentration.
   
   Values should increase monotonically down the column.
   
   UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/sec (LAB), kg/sm$^3$ (PVT-M)

2. The corresponding gas-water surface tension.
   
   UNITS: N/m (METRIC), lbf/in (FIELD), dynes/cm (LAB), N/m (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFU (see keyword TABDIMS).

This keyword is only relevant if the foam model is used with the functional gas mobility reduction model.

ECLIPSE 100

In ECLIPSE 100, this requires the FOAM option to be specified in the RUNSPEC section and FUNC to be set for item 2 of the FOAMOPTS keyword.

ECLIPSE 300

In ECLIPSE 300, the foam model is activated by using an additional water component COMPW to represent the foam surfactant CWTYPE. ECLIPSE 300 only supports the functional form of the gas mobility model, so this need not be specified.

See the "Foam model" in the ECLIPSE Technical Description for further information.

Examples

Example 1

With NTSFUN=1 and NSSFUN ≥ 5 and Metric units:

```
FOAMFST
---
  -- Conc  ST
  -- kg/sm3  N/m
---
  0.0   0.01
  10.0  0.005
  15.0  0.002
  20.0  0.001
  25.0  0.0001 /
```

Example 2

With NTSFUN=1 and NSSFUN ≥ 5 and Field units:

```
FOAMFST
---
  -- Conc  ST
  -- lb/stb  lbf/in
---
  0.0  0.00005?
```
Example 3

With NT SFUN=1 and NSSFUN ≥ 5 and Lab units:

<table>
<thead>
<tr>
<th>Conc</th>
<th>ST</th>
</tr>
</thead>
<tbody>
<tr>
<td>gm/scc</td>
<td>dyne/cm</td>
</tr>
<tr>
<td>0.00</td>
<td>10.0</td>
</tr>
<tr>
<td>0.01</td>
<td>5.0</td>
</tr>
<tr>
<td>0.015</td>
<td>2.0</td>
</tr>
<tr>
<td>0.02</td>
<td>1.0</td>
</tr>
<tr>
<td>0.025</td>
<td>0.1</td>
</tr>
</tbody>
</table>

where the different sets of values are intended to be illustrative of the different orders of magnitudes associated with the different input unit systems.
FOAMFSW  Gas mobility reduction dependence upon water saturation

The data comprises NTSFUN (see keyword TABDIMS) tables defining the parameters which control the gas mobility reduction due to foam as a function of water saturation. Each table consists of one row of data containing two items which should be terminated with a slash (/):

1. The limiting water saturation below which the foam ceases to be effective.
2. A weighting factor which controls the sharpness in the change of mobility.

DEFAULT: 1.0

The first item in the first table must be specified. The second item may be defaulted. If the second or subsequent tables are completely defaulted, the associated parameters will be copied from the previous table.

This keyword is only relevant if the foam model is used with the functional gas mobility reduction model.

ECLIPSE 100

In ECLIPSE 100 this requires the FOAM option to be specified in the RUNSPEC section and FUNC to be set for item 2 of the FOAMOPTS keyword.

ECLIPSE 300

In ECLIPSE 300 the foam model is activated by using an additional water component COMPW to represent the foam surfactant CWTYPE. ECLIPSE 300 only supports the functional form of the gas mobility model, so this need not be specified.

In both simulators, the FOAMFSC keyword must be specified with the functional gas mobility reduction model. See "Foam model" in the ECLIPSE Technical Description for further information.

For the functional model of gas mobility reduction, the mobility reduction factor $M_{rf}$ is expressed in the form:

$$M_{rf} = \frac{1}{1 + (M_r \cdot F_s \cdot F_w \cdot F_o \cdot F_c)}$$

where

$M_r$ is the reference mobility reduction factor (FOAMFRM keyword).

$F_s$ is the mobility reduction factor dependence upon surfactant concentration (FOAMFSC keyword).

$F_w$ is the mobility reduction factor dependence upon water saturation (this keyword).

$F_o$ is the mobility reduction factor dependence upon oil saturation (FOAMFSO keyword).

$F_c$ is the mobility reduction factor dependence upon capillary number (FOAMFCN keyword).

The mobility reduction factor functional dependence upon water saturation $F_w$ is expressed in the form:

$$F_w = 0.5 + \frac{\tan( f_w \left( S_w \cdot S_w^l \right) \pi)}{\pi}$$

where

Keywords
$S_w$ is the water saturation.

$S_w^f$ is the limiting water saturation below which the foam ceases to be effective.

$f_w$ is a weighting factor which controls the sharpness in the change in mobility.

The contribution of this component to the overall gas mobility reduction is determined by the limiting water saturation, below which the foam ceases to be effective, and the value of the weighting factor which controls the sharpness of the change about the transition point $S_w = S_w^f$. This keyword is optional. If this keyword is not specified the component of the overall mobility reduction factor which models the functional dependence upon water saturation will default to 1.0. See also the FOAMFCN, FOAMFSC, FOAMFRM and FOAMFSO keywords.

The water saturation functional mobility reduction term is only applied in grid blocks which contain foam with a concentration which is greater than or equal to the minimum concentration specified by item 3 of the FOAMFSC keyword.

**Example**

With NTSFUN=5

```
FOAMFSW
  -- S_w  f_w
     0.3   1E3 /
    0.31  1E3 /
   / 0.32  2e4 /
   / 0.32  2e4 /
```
FOAMMOB  Gas mobility reduction data

This keyword supplies tables of the gas phase mobility reduction factor as a function of foam concentration. The keyword is obligatory in runs using the foam model as described in "Foam model" in the ECLIPSE Technical Description.

The data comprises NTPVT (see keyword TABDIMS) tables of mobility reduction factors. Each table consists of two columns of data, terminated with a slash (/).

1. The foam concentration.
   The values should increase monotonically down the column.

   If item 1 of FOAMOPTS is set to ‘GAS’:
   UNITS: kg/sm$^3$ (METRIC), lb/Mscf (FIELD), gm/scc (LAB)

   If item 1 of FOAMOPTS is set to ‘WATER’:
   UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scc (LAB)

2. The corresponding gas mobility reduction factor
   There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see keyword TABDIMS).

Example

With NTPVT=1 and NPPVT ≥5:

<table>
<thead>
<tr>
<th>FoamMob</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Concentration</td>
</tr>
<tr>
<td>0.00</td>
<td>1.0</td>
</tr>
<tr>
<td>0.01</td>
<td>0.2</td>
</tr>
<tr>
<td>0.015</td>
<td>0.1</td>
</tr>
<tr>
<td>0.02</td>
<td>0.09</td>
</tr>
<tr>
<td>0.025</td>
<td>0.09 /</td>
</tr>
</tbody>
</table>

Keywords

828
**FOAMMOBP  Pressure dependence of foam mobility reduction**

This keyword is optional in runs using the foam model, and supplies tables of data to describe the pressure dependence of the foam mobility reduction factor. See "Foam model" in the ECLIPSE Technical Description for further information.

The data comprises NTPVT (see keyword TABDIMS) tables of mobility data. Each table consists of two columns of data, terminated with a slash (/).

1. The oil phase pressure.
   - The values should increase monotonically down the column.
   - UNITS: barsa (METRIC), psia (FIELD), atma (LAB)

2. The corresponding pressure modifier on the foam mobility reduction factor, $M_p(P)$. The values are usually between zero and 1.
   - 0.0 leaves the mobility reduction factor unaffected
   - 1.0 returns the gas mobility to the pure gas mobility.

   $$MP = (1 - M(C_{foam})) \cdot M_p(P) + M(C_{foam})$$

   where
   - $MP$ is the mobility reduction factor with the pressure effect
   - $M(C_{foam})$ is the original reduction factor as a function of foam concentration (FOAMMOB keyword)
   - $M_p(P)$ is the pressure dependency function
   - $P$ is the oil pressure.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see keyword TABDIMS).

**Example**

With NTPVT=1 and NPPVT≥3:

```
FOAMMOBP
--
-- Pressure      Pressure modifier
--
3000            0.0
5000            0.0
10000           0.8  /
```
**FOAMMOBS  Shear dependence of foam mobility reduction**

This keyword is optional in runs using the foam model, and supplies tables of data to describe the shear dependence of the foam mobility reduction factor. See "Foam model" in the *ECLIPSE Technical Description* for further information.

The data comprises NTPVT (see keyword TABDIMS) tables of mobility data. Each table consists of two columns of data, terminated with a slash (/).

1. The gas phase flow velocity.
   - The values should increase monotonically down the column.
   - The first entry should be 0.0.
   - **UNITS:** m/day (**METRIC**), ft/day (**FIELD**), cm/hr (**LAB**)

2. The corresponding shear modifier on the foam mobility reduction factor, $M_s(v)$.
   - The values are usually between zero and 1.
   - 0.0 leaves the mobility reduction factor unaffected
   - 1.0 returns the gas mobility to the pure gas mobility.

$$MF = (1-MP)M_s(v) + MP$$

where

- **MF** is the final mobility reduction factor
- **MP** is the mobility reduction factor including the pressure effect (from the **FOAMMOB** and **FOAMMOBP** keywords)
- $M_s(v)$ is the shear dependency function
- $v$ is the gas velocity.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see keyword TABDIMS).

**Example**

With **NTPVT=1** and **NPPVT >=4**:

```
FOAMMOBS
---
  Velocity  Shear modifier
---
  0.0       0.0
  0.02      0.0
  0.04      1.0
  0.06      1.0 /
```
FOAMOPTS  Foam Model options

This keyword consists of two items which define the form of the foam model:

1. Transport phase for foam surfactant.
   - GAS  Specifies that the foam is transported in the gas phase.
   - WATER Specifies that the foam is transported in the water phase.
   DEFAULT: GAS

2. Gas mobility reduction factor model.
   - TAB  Specifies that the gas mobility reduction factor is to be modeled using the tabulated functions of gas mobility reduction versus concentration (FOAMMOB keyword) and optionally versus shear (FOAMMOBS keyword) and pressure (FOAMMOBP keyword). This tabular form of modeling the gas mobility reduction factor is available with either GAS or WATER specified as the transport phase for foam.
   - FUNC Specifies that the gas mobility reduction factor is to be modeled using the functional form in which the mobility reduction factor is defined to be a combination of the mobility reduction factors due to a reference mobility factor (FOAMFRM keyword) and a function of surfactant concentration (FOAMFSC keyword), water saturation (FOAMFSW keyword), oil saturation (FOAMFSO keyword) and capillary number (FOAMFCN and FOAMFST keywords). This functional form of modeling the gas mobility reduction factor is only available if the transport phase for foam is specified to be WATER.
   DEFAULT: TAB

This keyword is only relevant if the FOAM option has been specified in the RUNSPEC section.

See "Foam model" in the ECLIPSE Technical Description for further information.

Example

```plaintext
FOAMOPTS
  WATER FUNC /
```
FOAMROCK  Specifies the foam-rock properties

The data comprises NTSFUN (see keyword TABDIMS) tables specifying the rock properties which are required for the foam model. Each table requires the following items of data, terminated by a slash (/).

1. The adsorption index to be used for this rock type.
   - Possible values for the adsorption index are 1 or 2.
   - If a value of 1 is selected, then the foam adsorption isotherm is retraced whenever the local foam concentration in the solution decreases.
   - If a value of 2 is selected, then no foam desorption may occur.
   - DEFAULT: 1

2. The mass density of this rock type at reservoir conditions.
   - This quantity is used in the calculation of the foam loss due to adsorption.
   - UNITS: kg/m^3 (METRIC), lb/ft^3 (FIELD), gm/cc (LAB)

Note that if the Polymer or Surfactant Models are being used, then the mass density may be ignored. The mass density will be taken from the first keyword present in the following hierarchy: PLYROCK, SURFROCK, FOAMROCK.

See also keyword FOAMADS and "Foam model" in the ECLIPSE Technical Description for further information.

Note: When non-zero initial foam concentrations are specified via the SFOAM or TVDP keywords, the concentration specified in that keyword will be honored and extra adsorbed foam is added to the rock so that it is in equilibrium with the input fluid concentration.

Example

With NTSFUN=3:

```
FOAMROCK
  2 2381 /
  2 2342 /
  1 2520 /
```
FORMFEED  Sets the formfeed character in the print file

This keyword allows the user to specify how ECLIPSE should insert formfeeds in the Print and Run Summary files. The keyword is optional, the default being standard FORTRAN carriage control. The most appropriate place for the keyword is at the top of the dataset.

The keyword should be followed by a single integer, terminated by a slash (/), to select one of the following options:

1. Standard FORTRAN carriage control
2. Formfeed character, ASCII(12)
3. No formfeed

DEFAULT: 1 (standard FORTRAN carriage control)

Example

Set the formfeed to a formfeed character for a laserjet printer.

```
FORMFEED 2 /
```
FORMOPTS  Formulation options

The keyword allows the user control of some formulations options typically associated with compositional effects in ECLIPSE 300. These options should only make very small differences to the simulator output but in some cases may significantly affect the performance of the simulation. In future versions of the program the default options may change, with this keyword available for back compatibility.

The keyword should be followed by a list of options required. The default is for all these options to be switched off.

The data should be terminated by a slash (/).

**Note:** From the 2002A release onwards HCSCAL has been turned on by default. Back compatibility can be enforced by using the NOHCSCAL keyword.

---

**HCSCAL**  If the HCSCAL option is active an extra scaling of the hydrocarbon relative permeabilities is performed. In a compositional model, oil can change to gas without passing through a two-phase state. The hydrocarbon relative permeabilities are interpolated to give a smooth transition between the two states (see "Near critical oil and gas relative permeabilities" in the ECLIPSE Technical Description). However, if the gas relative permeability at connate water differs from the oil relative permeability (note that Krog has to equal Krow at connate water) then a discontinuity can occur in the hydrocarbon relative permeability as a system becomes two phase. This discontinuity can be avoided by making Krg equal Kro at connate water, which is not an unreasonable assumption. Alternatively, the HCSCAL option scales Krg near the critical point in an analogous manner to the hydrocarbon relative permeability, again avoiding a discontinuity.

**NOHCSCAL**  Turns off the HCSCAL option

The HCSCAL option will be turned off automatically if the ODD3P relative permeability and capillary pressure hysteresis model is selected or if the Surfactant Model is active (see "Surfactant model" in the ECLIPSE Technical Description). This is equivalent to specifying NOHCSCAL.

**SINGLEP**  The SINGLEP option modifies the densities used in the hydrostatic term for flow between two cells when the fluid is a single phase hydrocarbon. By default the code labels a single phase hydrocarbon as oil or gas, depending on the fluid critical temperature as estimated by the Li correlation. The density for each phase is then taken as the saturation weighted density of the two cells between which a flow is being calculated. In the case of a single phase hydrocarbon this leads to discontinuities as a cell changes state, whereas physically we know the fluid is a continuous single phase. The SINGLEP option treats a single phase hydrocarbon as just that, with the density taken to be the (saturation weighted) hydrocarbon density without worrying if the hydrocarbon is an oil or a gas. Note that two phase behavior is unaltered.

The SINGLEP option cannot be used in thermal runs.

**NOPCSPT**  The NOPCSPT (No Pc Split) option modifies how the gas-oil capillary pressure is treated. By default ECLIPSE 300 (only in the compositional mode) splits the gas-oil capillary pressure between the oil and gas phases based on the saturation:

\[ P_{oil} = P - P_{cog} \times \left( \frac{S_g}{S_o + S_g} \right) \]
\[ P_{gas} = P + P_{cog} \times \left( \frac{S_o}{S_o + S_g} \right) \]

Essentially, the solution variable pressure, \( P \), is the hydrocarbon pressure rather than the phase pressure for either gas or oil. This prevents many of the discontinuities associated with the single phase transition of oil to gas; given a phase relabeling the phase pressure will remain the same. However because of the upwinding of mobilities in the flow, discontinuities can still occur in the flows.

If the `NOPCSPT` option is used then the capillary pressure is assumed to be associated with the gas phase:

\[ P_{oil} = P \]

\[ P_{gas} = P + P_{cog} \]

This option is not recommended in compositional cases where the fluid is likely to reach a state near to the critical point.

Note that this is the assumption in ECLIPSE 100.

The `NOPCSPT` option will be turned on automatically if the `ODD3P` relative permeability and capillary pressure hysteresis model is selected.

**Example**

Use the single phase density options, but leave the default capillary pressure treatment.

```
FORMOPTS
SINGLEP /
```
**FREEZEPC**  Freeze all capillary pressures in a run

This keyword selects the FREEZEPC solution option. This is useful in IMPES or AIM runs which involve large capillary pressure. These effectively turns saturation variation into pressure variation, and can cause instability. This type of instability is not related to well or cell throughputs, and can cause an instability to build up from a quiescent initial solution.

FREEZEPC is not used by default in any mode, and the rate at which capillary pressures may change is limited in IMPES and AIM modes (by the **DPCDT** keyword); this is usually sufficient to prevent stability problems.

FREEZEPC contains no items.
**FRICTION**

**Enables the wellbore friction option**

This keyword must be entered to enable the wellbore friction option (see "Wellbore friction option" in the *ECLIPSE Technical Description*). It specifies the maximum number of wells requiring the wellbore friction calculation, and (for multilateral wells) the maximum number of branches that can join within any single grid block connection.

The keyword should be followed by two items of data, terminated by a slash (/):

1. **NWFRIC** The maximum number of wells requiring the wellbore friction calculation
   
   DEFAULT: 0

2. **NWFRIB** The maximum number of branches, or flow streams, that may join together at a single connection (including the main stem). For example, if no more than one side branch joins the main stem within any grid block connection, the number should be set to 2.
   
   DEFAULT: 1 (describing a standard well, which has only a main stem)

**Example**

```
FRICTION
  3  2 /
```
FULLIMP

The fully implicit solution option

This keyword selects the Fully Implicit Solution option.

See "Formulation of the equations" in the ECLIPSE Technical Description.

FULLIMP contains no items.

In ECLIPSE 300. This is often required for runs with very high flow rates, such as dual porosity modeling and radial runs. This is the default for BLACKOIL, Thermal and Coal Bed Methane simulations.

In ECLIPSE 100. This is the default solution option in ECLIPSE 100.
**FVST**

**Specify miscibility variation with surface tension**

This provides the variation of the relative permeability miscibility factor $F_K$ with surface tension. $F_K$ is the weight factor for the **immiscible** contribution (see equation 3.73), and should increase with increasing surface tension.

If **FVST** is not entered the miscibility factor will be obtained using the Coats expression:

$$F_K = \left(\frac{\sigma}{\sigma_0}\right)^N$$  \hspace{1cm} \text{Eq. 3.73}

where $N$ is the exponent set using **MISCEXP** and defaulted to 0.25.

In either case $F_K$ is used to define the oil and gas relative permeability curves:

$$K_r = F K_r^{\text{imm}} + (1-F) K_r^{\text{mis}}$$  \hspace{1cm} \text{Eq. 3.74}

The two contributions are scaled so that they both have the same critical saturation, given by $F S_{cr}^{\text{imm}}$.

**FVST** avoids the use of both $P$ and $\sigma_0$ in relative permeability calculations. $\sigma_0$ is still used in modifying capillary pressures:

$$P_{cog}^{\text{corr}} = \left(\frac{\sigma}{\sigma_0}\right) P_{cog}$$  \hspace{1cm} \text{Eq. 3.75}

The number of tables should be equal to the number of pressure tables in the run - one is required for each pressure table (**PVTNUM** region).

Items in each table:

1. The surface tension.

   \text{UNITS: dynes/cm (**METRIC**), dynes/cm (**FIELD**), dynes/cm (**LAB**), dynes/cm (**PVT-M**)}

2. The miscibility factor $F_K$.

   \text{UNITS: dimensionless}

A table is terminated with a slash (/).

**Example**

```
FVST
0  0.0
5  0.25
10 0.5
20 1.0
40 1.0 /
```
**FWELLS**

Fractured well completion option

If this keyword is entered in the **RUNSPEC** section, it is possible to request fracture completions (FX and FY) in the **WELLCOMP** keyword (item 12) or the **COMPDAT** keyword (item 13).

The keyword **FWELLS** contains no items.
This section contains an alphabetic listing of keywords starting with the letter G.
GADJUST

**Adjust diagonal permeability tensor**

This keyword specifies that the permeability tensor is assumed to be input as a diagonal tensor on the basis defined by the cell tangent vectors. It is supplied only for FrontSim compatibility.

This is equivalent to using item 2 in the `TBASIS` keyword as long as the off-diagonal coefficients, for example `PERMXY`, are zero.

See "Tensor permeability" in the *ECLIPSE Technical Description* for further information.

**Example**

```
GADJUST
```

---

**Keywords**

842
GADVANCE  Assign advance import gas to a group

This keyword assigns a supply of advance import gas to a group supplying gas for reinjection (see keywords GINJGAS, WINJGAS, GRUPINJE and WELLINJE). When gas from this group is reinjected, the advance gas will be used for reinjection prior to the gas produced by this group from the reservoir. If the reinjection requirements can be met by the advance gas stream alone, then no produced gas is reinjected and all of it remains available for fuel and/or sales usage. If the advance and available produced gas streams together are not sufficient to satisfy the reinjection requirements, make-up gas, if specified for the injection group or well, is used to make up the shortfall.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated by a slash (/). The set of records must be ended by a blank record, containing only a slash.

1. Group name or group name root,
   or FIELD (for import to the Field group)
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. The name of the well stream (defined using the WELLSTRE keyword) which defines the composition of the advance gas.

3. The maximum rate at which advance gas can be supplied.
   UNITS: sm³/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm³/day (PVT-M)

By setting OPTIONS3 item 30, an alternative ‘mixing tank’ treatment may be invoked, where the advance gas is mixed with the produced gas and gas from this ‘mixing tank’ is reinjected. The fraction of advance gas in this mixture is equal to the ratio of the specified advance gas rate (item 3) to the sum of the specified advance gas rate and the available gas production rate. With this option, the advance gas is not used in preference to the produced gas.

If advance gas is used, the injection calculation may have to be iterated to match the injection gas composition with the injection rate (see keyword GCONTOL).

Note: Advance gas is assigned to the source group supplying the gas, while make-up gas is assigned to the group or well having the injection target. Also, advance gas cannot be used for fuel and sales; it can only be used for injection.

Example

Advance gas is assigned to group GA1 at a maximum rate of 10000 Mscf/day.

```
GADVANCE
  GA1 STR1 10000 /
```

Keywords 843
GAS

Run contains GAS

This keyword indicates that a run contains gas. This may be specified whenever a gas phase exists or could exist.

In ECLIPSE 300, this keyword is automatically supplied for any compositional mode run.

The keyword GAS contain no items.
**GASADCO**  **Initial coal gas adsorbed concentration**

This keyword specifies the initial coal gas concentration for each grid cell. The keyword should be followed by \( N_x \cdot N_y \cdot N_z \cdot N_c \) values. The data field should be terminated by a slash (/).

**UNITS:** \( \text{sm}^3/\text{m}^3 \) (METRIC), Mscf/ft\(^3\) (FIELD), scc/cc (LAB)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to \( N_{DIVIX} \)), followed by the Y axis index (from 1 to \( N_{DIVIY} \)), and then the Z axis index (from 1 to \( N_{DIVIZ} \)) for each component. \( N_{DIVIX} \), \( N_{DIVIY} \) and \( N_{DIVIZ} \) are specified in RUNSPEC keyword **DIMENS**. Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*).

The keyword is not compatible with the instant adsorption model (**CBMOPTS**).

Only values for coal cells are used. The keyword resets the coal concentration but \( YMF \) values still need to be specified. The \( YMF \) values are however only used if **GASADEC** or **GASSATC** are used.

Note that the keyword cannot be used together with **GASCONC**.

**Example**

With \( N_{DIVIX}=5, N_{DIVIY}=4, N_{DIVIZ}=10 \) and \( NCOMPS=2 \)

```plaintext
GASADCO
   --matrix fracture
   100*86.0 100*0.0 --component 1
   100*43.0 100*0.0 --component 2/
```

---

**Keywords**

845
GASADEC  Initial equilibrium coal gas concentration

This keyword specifies the initial equilibrium coal gas concentration for each grid cell. The keyword should be followed by $N_x \cdot N_y \cdot N_z \cdot N_c$ values. The data field should be terminated by a slash (/).

UNITs: $\text{sm}^3/\text{m}^3$ (METRIC), Mscf/ft$^3$ (FIELD), scc/cc (LAB)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and then the Z axis index (from 1 to NDIVIZ) for each component. NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*).

This information is used to scale the Langmuir isotherm, as specified by the LANGMEXT or the LANGMUIR keyword, at the initial pressure and composition in the reservoir. This means that a non-zero initial composition needs to be specified for each component in order to perform the scaling. If the component is not present initially, the keyword GASADCO can be used to set the initial coal gas concentration.

Only values for coal cells are used. Note that the composition ($\text{YMF}$) and pressure value of the coal cell is used and not the connected pore volume (fracture) composition and pressure.

Note that the keyword cannot be used together with LANGMULT, LANGMULC or GASSATC.

Example

With NDIVIX=4, NDIVIY=5, NDIVIZ=10 and NCOMPS=2

```
GASADEC
   --matrix fracture
   100*86.0 100*0.0 --component 1
   100*43.0 100*0.0 --component 2/
```
GASBEGIN

Starts set of keywords for annual scheduling file

GASBEGIN denotes the start of the set of keywords to be placed in the annual scheduling file, for use with the gas field operations model (see "Gas field operations model" in the ECLIPSE Technical Description). This keyword has no associated data.

All keywords entered between the keywords GASBEGIN and GASEND are placed on the Annual Scheduling File. If the GASBEGIN - GASEND pair are subsequently entered again, the contents of the Annual Scheduling file is replaced with the new set of intervening keywords. The Annual Scheduling file can be emptied by following immediately with GASEND.

The annual scheduling file

The annual scheduling file is designed for use with the Gas Field Operations Model, to enable the engineer to change well controls and request printed reports during the year, at times other than the start or end of a contract period.

The keywords GASYEAR and GASPERIO advance the simulation through one or more complete contract years or contract periods. Well operations can be performed in the usual manner at the end of a contract period, and a printed report is generated at the end of each contract period depending on the RPTSCHED switches. But in order to perform well operations or request printed reports during a contract period, an annual scheduling file must be set up.

To set up an annual scheduling file, enter the keywords defining the required operations before the GASYEAR or GASPERIO keyword on the input data file, bracketed with the keywords GASBEGIN and GASEND. When ECLIPSE reads the keyword GASBEGIN during the simulation, it copies the subsequent keywords up to GASEND to a separate file that it sets up automatically. It does not read the keyword data and perform the appropriate actions until it reads the GASYEAR or GASPERIO keyword. The keywords in the annual scheduling file is then read and processed each year while the simulation is being advanced with the GASYEAR or GASPERIO keyword. The contents of the file are ignored if the simulation is being advanced with a DATES, TIME or TSTEP keyword.

Note: Please ensure that the keyword data (but not the keyword itself) is indented by at least one space from the left hand margin.

When the GASYEAR or GASPERIO keyword instructs ECLIPSE to advance the simulation through a contract year or period, ECLIPSE reads the keyword data on the annual scheduling file and performs the required actions at the appropriate times. The actions can be requested at the beginning of each month named with the keyword GASMTH. If the contract period is solved in two or more passes (to work out the DCQ), the Annual Scheduling file is rewound and read from the beginning of the period again at the start of each pass.

Once an Annual Scheduling file has been set up, ECLIPSE reads its keywords and perform the required actions during every subsequent contract period specified with a GASYEAR or GASPERIO keyword. Thus if the same actions are to be performed every year, the Annual Scheduling file need only be set up once. But if the actions must be changed during the simulation, a new annual scheduling file must be set up. The new file is set up in the same way as before; bracketing the required keywords with GASBEGIN and GASEND. The contents of the original file are erased, and replaced with the new set of keywords. If no more actions are required, the Annual Scheduling file can be emptied by following GASBEGIN immediately with GASEND.

In a restart run, the annual scheduling file must be set up again using GASBEGIN and GASEND; it is not automatically carried over in the Restart file. If you forget to do this, ECLIPSE issues an error message, as
it detects that the base run had an Annual Scheduling file but there was no such file in the restart run. If you no longer need an Annual Scheduling file in the restart run, you must still set up an empty one.

**Allowed keywords for the annual scheduling file**

Only a subset of the ECLIPSE keywords is allowed on the Annual Scheduling file. These are:

<table>
<thead>
<tr>
<th>Restriction</th>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td><strong>BRANPROP</strong></td>
<td>to change branch properties when using the extended network model</td>
</tr>
<tr>
<td>E100</td>
<td><strong>DEBUG</strong></td>
<td>to change debug switches.</td>
</tr>
<tr>
<td>E300</td>
<td><strong>DEBUG3</strong></td>
<td>to change debug switches.</td>
</tr>
<tr>
<td>Compulsory</td>
<td><strong>GASEND</strong></td>
<td>to mark the end of the annual scheduling file.</td>
</tr>
<tr>
<td></td>
<td><strong>GASMONT</strong></td>
<td>to indicate the time of the next set of operations.</td>
</tr>
<tr>
<td></td>
<td><strong>GCONPRI</strong></td>
<td>to change group limits under prioritization group control.</td>
</tr>
<tr>
<td></td>
<td><strong>GCONFPROD</strong></td>
<td>to change group limits and group guide rates under guide rate group control.</td>
</tr>
<tr>
<td>E100</td>
<td><strong>GCONSUMP</strong></td>
<td>to change group gas consumption rates.</td>
</tr>
<tr>
<td>E300</td>
<td><strong>GRUPFUEL</strong></td>
<td>to change group gas fuel rates.</td>
</tr>
<tr>
<td></td>
<td><strong>GRUPNET</strong></td>
<td>to change the VFP table or ALQ value on network branches, or to change the pressure of an existing fixed pressure node.</td>
</tr>
<tr>
<td></td>
<td><strong>GRUPTARG</strong></td>
<td>to change individual group rate limits or guide rates.</td>
</tr>
<tr>
<td></td>
<td><strong>GRUPTREE</strong></td>
<td>to change the group hierarchy.</td>
</tr>
<tr>
<td></td>
<td><strong>GSATINJE</strong></td>
<td>to change injection rates of satellite groups.</td>
</tr>
<tr>
<td></td>
<td><strong>GSATPROD</strong></td>
<td>to change production rates of satellite groups.</td>
</tr>
<tr>
<td>E100</td>
<td><strong>NCONSUMP</strong></td>
<td>to change gas consumption at nodes in the extended network model.</td>
</tr>
<tr>
<td></td>
<td><strong>NEXTSTEP</strong></td>
<td>to limit the length of the next timestep, for example to reduce it after some wells have been opened.</td>
</tr>
<tr>
<td></td>
<td><strong>NODEPROP</strong></td>
<td>to change nodal properties in the extended network model.</td>
</tr>
<tr>
<td>E300</td>
<td><strong>RPTPRINT</strong></td>
<td>to change switches for printed reports.</td>
</tr>
<tr>
<td>E100</td>
<td><strong>RPTSCHEDED</strong></td>
<td>to change switches for printed reports. The following report switches will be ignored here:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• ’RESTART’ - restart files are only written at the end of a contract period</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• ’CPU’ - the timer switch setting must not be altered within the annual scheduling file.</td>
</tr>
<tr>
<td>E300</td>
<td><strong>TSCRIT</strong></td>
<td>to modify timestepping criteria.</td>
</tr>
<tr>
<td></td>
<td><strong>TUNING</strong></td>
<td>to modify tuning parameters, for example maximum step size.</td>
</tr>
<tr>
<td></td>
<td><strong>WEFAC</strong></td>
<td>to change well efficiency factors.</td>
</tr>
<tr>
<td></td>
<td><strong>WELDEBUG</strong></td>
<td>to change well debug switches.</td>
</tr>
<tr>
<td></td>
<td><strong>WELOPEN</strong></td>
<td>to open or close wells or connections.</td>
</tr>
<tr>
<td></td>
<td><strong>WELPRI</strong></td>
<td>to change well priorities for prioritization group control.</td>
</tr>
<tr>
<td></td>
<td><strong>WELTARG</strong></td>
<td>to change well operating limits.</td>
</tr>
</tbody>
</table>
Restriction | Keyword | Description
--- | --- | ---
 | WGRUPCON | to change well guide rates for guide rate group control.

**Example**

For a contract year beginning on 1st September

Start the set of keywords to be placed on the annual scheduling file

```
GASBEGIN
```

Remove the summer downtime factor on well P2

```
WEFAC
  P2 1.0 /
```

Set output switches so that only a well report will be generated

```
RPTSCHED
  'WELLS=2' 'SUMMARY=2' 'WELSPECS' /
```

Advance the simulation to 1st Jan., and request a report

```
GASMONTH
  JAN YES /
```

Advance the simulation to 1st June, without a report

```
GASMONTH
  JUN NO /
```

From 1st June, impose a summer downtime factor on well P2

```
WEFAC
  P2 0.8 /
```

Restore pressure and saturation switches for the end of contract year report

```
RPTSCHED
  'PRES' 'SWAT' 'SGAS' 'WELLS=2' 'SUMMARY=2'
  'WELSPECS'
```

End the annual scheduling file

```
GASEND
```

Simulate 3 contract years using this annual scheduling file

```
GASYEAR
  3  100000 ACQ /
```

Subsequent years do not have the summer downtime factor on well P2, but still require a report on 1st January.
First remove the summer downtime factor imposed in the last contract

```
WEFAC
  P2  1.0 /
```

Set up a new annual scheduling file with just the winter report

```
GASBEGIN
RPTSCHED
  'WELLS=2' 'SUMMARY=2' 'WELSPECS' /
GASMONTH
  JAN  YES /
RPTSCHED
  'PRES' 'SWAT' 'SGAS' 'WELLS=2' 'SUMMARY=2'
  'WELSPECS' /
GASEND
```

Simulate 4 more contract years using the new annual scheduling file

```
GASYEAR
  4  1*  ACQ /
END
```
### GASCCMP

**GASSATC or GASCONC component number**

This keyword is used to specify a component number to be used with the **GASSATC** or the **GASCONC** keyword. If there is only one component (NCOMPS=1) the keyword is optional.

**Example**

```
GASCCMP
2 /
```

---

**Keywords**

851
**GASCONC**

**Initial coal gas concentration**

The keyword should be followed by one real number for every matrix grid block, specifying the initial gas concentration in the coal. The data field should be terminated by a slash (/).

**UNITS:** \( \text{sm}^3/\text{m}^3 \) (METRIC), \( \text{Mscf}/\text{ft}^3 \) (FIELD), \( \text{scc}/\text{cc} \) (LAB)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to \( \text{NDIVIX} \)), followed by the Y axis index (from 1 to \( \text{NDIVIY} \)), and finally the Z axis index (from 1 to \( \text{NDIVIZ} \)). \( \text{NDIVIX}, \text{NDIVIY} \) and \( \text{NDIVIZ} \) are specified in **RUNSPEC** keyword **DIMENS**. Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*). Data need only be supplied for the matrix cells, that is the upper half of the grid.

Both **GASCONC** and **GCVD** (ECLIPSE 100) are entirely optional. If neither of these keywords is present, the coal gas concentration is taken as the equilibrium concentration for the corresponding fracture pressure.

The initial coal gas concentration should not be set above the initial equilibrium value. ECLIPSE issues a warning if it is.

**ECLIPSE 100**

See the **GCVD** keyword.

**ECLIPSE 300**

**GASCONC** can only be used to specify the initial coal gas concentration for one component which is given by the keyword **GASCCMP**. The keyword **GASADCO** can be used for the general case. Note that the keyword is not compatible with the instant adsorption model (**CBMOPTS**). Only values for coal cells are used. The keyword resets the coal concentration but \( \text{YMF} \) values still need to be specified. The \( \text{YMF} \) values are however only used if **GASADEC** or **GASSATC** are used. Note that the keyword cannot be used together with **GASADCO**.

**Example**

With \( \text{NDIVIX}=8, \text{NDIVIY}=6, \text{NDIVIZ}=10 \) specified in **RUNSPEC** keyword **DIMENS**:  

```
GASCONC
  48*89.1 48*87.3 48*85.1 48*85.0 48*85.0 /
```

**Keywords**

852
**GASEND**

**Ends set of keywords for annual scheduling file**

GASEND denotes the end of the set of keywords to be placed in the annual scheduling file, for use with the gas field operations model. This keyword has no associated data.

See keyword GASBEGIN for a more detailed description of the Annual Scheduling file.

<table>
<thead>
<tr>
<th>Keywords</th>
</tr>
</thead>
<tbody>
<tr>
<td>ECLIPSE 100</td>
</tr>
<tr>
<td>ECLIPSE 300</td>
</tr>
<tr>
<td>SPECIAL</td>
</tr>
<tr>
<td>RUNSPEC</td>
</tr>
<tr>
<td>GRID</td>
</tr>
<tr>
<td>EDIT</td>
</tr>
<tr>
<td>PROPS</td>
</tr>
<tr>
<td>REGIONS</td>
</tr>
<tr>
<td>SOLUTION</td>
</tr>
<tr>
<td>SUMMARY</td>
</tr>
<tr>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>

---

**853**
**GASFCOMP Automatic compressors for standard network option**

When the gas field operations model is run in conjunction with the standard network option, automatic compressors can be declared with this keyword. One compressor is allowed in each branch of the network. ECLIPSE switches on compression whenever necessary to enable the field to meet its gas production target. The compressors are turned off whenever the target rate decreases to see if the field can meet it without them. The automatic compressors are also turned on while calculating the field’s delivery capacity.

The Simple option for automatic compressors has just two states: all compressors on and all compressors off. Each compressor is defined by a single record of data.

The Extended option allows each compressor to have up to five levels of compression. The compressors will be turned on or switched to a higher compression level individually in a predefined order until the field can make its production target. The order in which these events occur is determined by an action sequence number. Compression levels with sequence number 1 are switched on first, followed by levels with sequence number 2, and so on in ascending order until either the rate target is met or all compressors are operating at their highest defined levels. The current operating level of each compressor can be written to the Summary file with the `SUMMARY` section keyword. All compressors are switched to their highest defined level while calculating the field’s delivery capacity. The Extended option requires a small amount of extra storage, which must be reserved by entering ‘YES’ in item 1 of keyword `GASFIELD` in the `RUNSPEC` section. Each compression level in each compressor is defined by a single record of data.

See "Gas field operations model” and “Network option” in the *ECLIPSE Technical Description* for further details.

*Note:* If you are using the Extended Network option (keyword `NETWORK` in the `RUNSPEC` section) then automatic compressors must be defined with the `NETCOMPA` keyword instead of `GASFCOMP`. The Extended Network compressors defined with keyword `NETCOMPA` have most of the functionality of the extended compressors defined with `GASFCOMP`.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root whose network pipeline contains the compressor

   The compressor associated with this group influences the pressure drop in the pipeline between this group and its parent group.

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. VFP table number to use when the compressor is operating at the level specified in item 5

   When the compressor is off, the table number specified in keyword `GRUPNET` is used.

   A zero or negative number means that the same VFP table number specified in `GRUPNET` is used.

   DEFAULT: 0

3. Artificial Lift Quantity to use when the compressor is operating at the level specified in Item 5

   When the compressor is off, the ALQ value specified in keyword `GRUPNET` will be used.

   UNITS: These depend on what the ALQ was meant to represent when the table was created. Refer to section “Record 6: ALQ values” in the description of keyword `VFPPROD`.

---

*Keywords* 854
4. Compressor’s gas consumption rate (used to fuel the compressor) when operating at the level specified in item 5

When the compressor is on, the extra gas consumption rate is added to the fixed value specified in GCONSUMP, if any.

UNITs: \(\text{sm}^3/\text{day (METRIC)}, \text{Mscf/day (FIELD)}, \text{scc/hr (LAB)}, \text{sm}^3/\text{day (PVT-M)}\)

DEFAULT: 0.0

5. The compression level that items 2 - 4 represent

An integer between 1 and 5. A number greater than 1 may be entered only if you are using the extended compressor option (see keyword GASFIELD in the RUNSPEC section).

DEFAULT: 1

6. The action sequence number at which the compressor switches to this compression level

An integer \(\geq 0\). A number greater than 1 may be entered only if you are using the extended compressor option (see keyword GASFIELD in the RUNSPEC section). A zero value makes this level inactive.

Two levels in the same compressor cannot share the same action sequence number.

DEFAULT: 1

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also the COMPOFF keyword.

**Examples**

**Example 1**

To place a single-level compressor in the pipeline from PLAT-A (simple compressor option):

```
GASFCOMP
-- GROUP  NEW VFP   NEW   GAS      COMP   SEQUENCE
-- NAME   TAB NO    ALQ   CONSUMP  LEVEL   NUMBER
PLAT-A    1*      50    1000   /       / 
```

**Example 2**

To declare a pair of two-level compressors in the pipelines from PLAT-A and PLAT-B, and have them turned on in sequence (extended compressor option):

```
GASFCOMP
-- GROUP  NEW VFP   NEW   GAS      COMP   SEQUENCE
-- NAME   TAB NO    ALQ  CONSUMP  LEVEL   NUMBER
PLAT-A    1*       50   1000     1         1     /       / 
PLAT-A    1*       100  1500     2         2     /       / 
PLAT-B    1*       50   1000     1         3     /       / 
PLAT-B    1*       100  1500     2         4     /       / 
```
GASFDEC

Decrement for FIELD gas contract

The GASFDEC keyword enables a monthly adjustment to be made to the field gas production rate for the second pass of the gas field operations model. The adjusted rate at which the field is produced is then DCQ × PROFILE - DECREMENT.

The GASFDEC can be used in conjunction with the GASFTARG keyword in which case the field is produced at the lesser of DCQ × PROFILE - DECREMENT and the rate specified by the GASFTARG data items.

UNITS: sm³/day (METRIC), Mcf/day (FIELD), scf/hr (LAB), sm³/day (PVT-M)

(ECLIPSE 100 only) If item 1 of DCQDEFN is set to ‘ENERGY’:

UNITS: kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB), kJ/day (PVT-M)

See "Gas field operations model" in the ECLIPSE Technical Description for further details.

The GASFDEC keyword is followed by 12 numbers, terminating with a slash (/). The decrements for defaulted months are set to zero.

Example

```
GASFDEC
-- JAN FEB MAR APR MAY JUN JUL AUG SEP OCT NOV DEC
 2*1000.0 3* 2*1000.0 3* 2*500.0 /
```
**GASFDELC**  Delivery capacity definition for Gas Field Operations Model

The delivery capacity is defined as the sales gas production rate that would ensue immediately after the removal of all group and field gas rate targets and limits, at current grid block conditions. The GASFDELC keyword may be used to specify whether or not any group or field oil or water rate limits, and any rate limits on the wells themselves, should also be removed when calculating the delivery capacity.

The delivery capacity is calculated at every timestep after the first entry of the SWINGFAC keyword which effectively turns this facility on.

See "Gas field operations model" in the ECLIPSE Technical Description for further details.

The keyword is followed by a single integer (either 1 or 2), terminated with a slash (/).

1. Remove only the group and field gas rate targets and limits.
   
   Honor all other limits, including any well rate and pressure limits.

2. Remove all well, group and field rate limits.

3. Honor only the well BHP and THP limits. (If the wells have drawdown limits, these will be honored too if YES has been entered in item 4 of keyword WELDRAW.)

If this keyword is not used, definition number 1 will be assumed.

*Note:* Definition 2 is not allowed when there are any contract groups using the FIX2 option in item 3 of keyword GDCQ, for multiple contract groups.

**Example**

```
GASFDELC
2 /
```
Enables special options in the Gas Field Operations Model

This keyword is required if the gas field operations model is to be used with:

- either the extended compressor option (see keyword GASFCOMP),
- or multiple contract groups (defined with keyword GSWINGF),
- or the accelerated iteration scheme for calculating the DCQ in the first pass of the contract period (see "Calculating the DCQ value" in the ECLIPSE Technical Description, and item 8 of keyword GASYEAR or item 9 of GASPERIO).

The keyword may be omitted if the gas field operations model is used without these options. (See "Gas field operations model" in the ECLIPSE Technical Description for further details.)

**Note:** The extended compressor option need not be requested when the extended network option is being used (see "The extended network model" in the ECLIPSE Technical Description). Compressors in the extended network option (defined with keyword NETCOMPA) contain most of the functionality of the extended compressor option.

The keywords should be followed by some or all of the following items to indicate whether these special options are going to be used, followed by a slash (/).

1. Flag to indicate whether or not the extended compressor option or multiple contract groups are required
   - **YES** The extended compressor option is required, and/or multiple contract groups are defined
   - **NO** Otherwise
   - **DEFAULT:** NO

2. Flag to indicate whether the accelerated iteration scheme for calculating the DCQ in the first pass of the contract period is to be used.
   - **YES** The accelerated iteration scheme is used.
   - **NO** The original (pre-96A) algorithm is used.
   - **DEFAULT:** NO

**Example**

```
GASFIELD YES YES /
```

Keywords

858
The GASFTARG keyword specifies a gas rate at which the field is to be produced in the second pass of the gas field operations model.

For months in which a rate is specified in the GASFTARG keyword the field is produced at the lesser of DCQ × PROFILE and the specified rate. For months in which GASFTARG is defaulted the field is produced at DCQ × PROFILE. The GASFTARG keyword can also be used in conjunction with the GASFDECR keyword.

UNITS: $\text{sm}^3/\text{day (METRIC)}$, Mscf/day (FIELD), scc/hr (LAB), $\text{sm}^3/\text{day (PVT-M)}$

In ECLIPSE 100 only. If item 1 of DCQDEFN is set to ‘ENERGY’:

UNITS: kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB), kJ/day (PVT-M)

See "Gas field operations model" in the ECLIPSE Technical Description for further details.

The GASFTARG keyword is followed by 12 numbers, terminating with a slash (/).

**Example**

<table>
<thead>
<tr>
<th>GASFTARG</th>
</tr>
</thead>
<tbody>
<tr>
<td>-- JAN FEB MAR APR MAY JUN JUL AUG SEP OCT NOV DEC</td>
</tr>
<tr>
<td>2<em>50000.0 3</em></td>
</tr>
<tr>
<td>2<em>20000.0 3</em></td>
</tr>
<tr>
<td>2*50000.0 /</td>
</tr>
</tbody>
</table>
GASMONTH  Sets month pointer in annual scheduling file

GASMONTH is used in the annual scheduling file of the gas field operations model, to set the time of the next set of operations specified by the subsequent keywords. It can also be used to request a printed report to be written at specified times each year.

See keyword GASBEGIN for a more detailed description of the annual scheduling file.

The GASMONTH keyword should be followed by two items, terminated with a slash (/):

1. Name of month - three letters
   - JAN, FEB, MAR, APR, MAY, JUN, JUL, AUG, SEP, OCT, NOV, DEC
   - The operations defined by the subsequent keywords will be performed at the beginning of the specified month.
   - (JLY is an acceptable alternative to JUL.)

2. Write a printed report? - YES or NO
   - (The word may be abbreviated, only the first character is significant)
   - YES  A report for the timestep ending at the start of the specified month will be written to the Print file. The contents will depend on the current values of the RPTSCHED or RPTPRINT switches.
   - NO  No report will be written to the Print file at this time.

   DEFAULT: NO

The GASMONTH keyword is only valid when bracketed by the keywords GASBEGIN and GASEND. If more than one GASMONTH keyword is present, they must be in the correct order within the year starting from the beginning of the first contract period.

Example

For a contract year beginning on the 1st September:

```
GASBEGIN
GASMONTH SEP Y /
```

(The GASMONTH keyword was not necessary here. The pointer is already at 1st September, the beginning of the contract year. A report is always printed at the end of each contract period. This keyword therefore has no effect here.)

Enter actions to be performed on 1st September.

```
Scheduling keywords...
```

Advance pointer to 1st December, the beginning of the winter quarter. No report is required.

```
GASMONTH DEC N
```

Enter actions to be performed on 1st December.
Advance pointer to 1\textsuperscript{st} March, the beginning of the spring quarter. Print a report for the timestep ending 1\textsuperscript{st} March, the end of the winter quarter.

```
GASMONTH
MAR Y /
```

Enter actions to be performed on 1\textsuperscript{st} March.

```
Scheduling keywords...
```

Advance pointer to 1\textsuperscript{st} June, the beginning of the summer quarter. No report is required.

```
GASMONTH
JUN N /
```

Enter actions to be performed on 1\textsuperscript{st} June.

```
Scheduling keywords...
```

End the annual scheduling file.

```
GASEND
```

\textit{Note:} In this example, the last valid GASMONTH date is AUG, as the contract year ends at the end of that month. A SEP entry is interpreted as the beginning of the contract year, and would be out of order at the end of the annual scheduling file.
**GASPERIO**  **Advances simulator over ‘n’ gas field contract periods**

The GASPERIO keyword can be used instead of DATES, TIME or TSTEP to advance the simulation over an integral number of contract periods, when running the gas field operations model (see "Gas field operations model" in the ECLIPSE Technical Description). The contract period is the time over which the DCQ must remain fixed until the next review; it may be a year, or any number of months that is a factor of 12 (that is 1, 2, 3, 4 or 6). The keyword GASYEAR may be used instead of GASPERIO if the contract period is a year.

The GASPERIO keyword activates two features of the model:

- Generation of the annual production profile from the profile factors set in keyword SWINGFAC or GSWINGF.
- Automatic reduction of the DCQ when necessary to obey the swing factor requirement, using the swing factors set in keyword SWINGFAC or GSWINGF.

The SWINGFAC or GSWINGF keyword must have been entered previously to activate the gas field operations model.

The keyword should be followed by some or all of these data items, terminated with a slash (/):

1. Number of contract periods to simulate.

   The contract period begins at the current date when the GASPERIO keyword is read. An error will be flagged if this is not the first day of a month. At the end of each contract period, ECLIPSE writes an output report and dumps a Restart file if required.

2. Number of months in a contract period.

   This must be 1, 2, 3, 4, 6 or 12.

   DEFAULT: 12

3. The initial DCQ (Daily Contracted Quantity).

   When there is a single gas supply contract on the FIELD (keyword SWINGFAC), the initial DCQ must be specified in the first entry of the GASPERIO keyword. It should be defaulted in subsequent entries, unless you wish to reset its value.

   When there are separate gas supply contracts on different groups (that is keyword GSWINGF has been used to activate the multiple contract group facility; see "Applying separate gas supply contracts to individual groups" in the ECLIPSE Technical Description), this item should be defaulted. Any value entered here is ignored. Instead, the initial DCQs of the contract groups must be set with keyword GDCQ before the GASPERIO keyword is read.

   UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

   *ECLIPSE 100 only*

   If item 1 of DCQDEFN is set to ‘ENERGY’:

   UNITS: kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB), kJ/day (PVT-M)

   DEFAULT: Current value of DCQ

4. Swing requirement for automatic reduction of the DCQ
PER or YES The field (or each contract group) must be able to produce at the DCQ multiplied by the monthly swing factor, for the whole contract period.

PRO The field (or contract group) must be able to raise its production rate from DCQ × profile to DCQ × swing for an instant at any time in the contract period.

PCQ The field (or contract group) must be able to produce at the DCQ multiplied by the monthly swing factor continuously until it has produced its contracted quantity (= no. of days in period × DCQ).

NO The swing requirement is not tested. The DCQ remains at its current value, even if the field (or contract group) cannot produce at its target rate.

Items 5 - 8 of this keyword are ignored.

DEFAULT: PER

If any option except NO is requested, each contract period is solved twice. The first pass is a test calculation, which has the purpose of deciding how much the DCQ needs to be reduced in order to obey the swing requirement for the current contract period. The second pass simulates the actual performance of the field, with the sales gas rate target set each month to the new value of the DCQ multiplied by the monthly profile factor.

PER is a suitable option for cases where the monthly swing factors are the same as the profile factors, as it ensures the production profile fits underneath the delivery capacity decline curve. But for cases where the swing factors exceed the profile factors, the PRO or PCQ options may be more suitable. For more discussion of these options, see "Interpretations of the swing requirement" in the ECLIPSE Technical Description.

When using the multiple contract groups facility (initiated with keyword GSWINGF), only the options PER and NO are allowed if there is more than one variable-DCQ contract group (see keyword GDCQ). If all contract groups except one have fixed DCQs, all the above options are allowed.

Note: If the GASPERIO keyword is used with the PCQ option and a 12 month contract period, the results may be marginally different to those obtained using the ACQ option in the GASYEAR keyword. This is because a 12 month contract period has either 365 or 366 days as opposed to the average 365.25 days used with the GASYEAR keyword.

5. Should the timesteps be limited so that each month starts with a new timestep? - YES or NO

(Yes the word may be abbreviated, only the first character is significant)

YES ECLIPSE automatically limits each timestep so that it does not go beyond the end of the current month. This ensures that there is at least one Summary file entry per month, and that the network (if using the Network option) is balanced at least once per month.

NO ECLIPSE only limits the timestep to the end of the month if the production rate target changes, or if there is an operation on the Annual Scheduling file. All other means of limiting the timestep (see keyword TUNING) still remain in force.

DEFAULT: YES

6. Limiting DCQ reduction factor, FLIM, allowed in a single iteration of the first pass of the contract year.
This restricts the amount by which the DCQ can be reduced in a single iteration of the first pass of the contract period. If the DCQ needs to be reduced to a fraction less than the value specified here, it is reduced only to this specified fraction (DCQ × FLIM) and the first pass of the contract period is solved again, subject to the upper limit in item 7 on the number of iterations.

If you are using the accelerated iteration scheme (this is always the case for ECLIPSE 300 and controlled by item 2 of RUNSPEC section keyword GASFIELD for ECLIPSE 100), it is not necessary to have a limiting reduction factor. Although a small value (for example 0.1) could be useful when the field or a contract group is getting close to the end of its life. This would stop the iteration immediately jumping to a DCQ of zero when it may still be able to operate with a small value of the DCQ.

ECLIPSE 100

If you are not using the accelerated iteration scheme, the DCQ reduction must be limited more strictly to obtain accurate results, to prevent it from substantially overshooting the correct value (it will not increase again). The nearer the reduction factor is to 1.0 the more accurate will be the result, but at the expense of requiring more iterations of the first pass to converge it.

RANGE: 0.0 to 0.99
DEFAULT: 0.0 (Allowing unlimited reduction)

See "Calculating the DCQ value" in the ECLIPSE Technical Description.

7. Anticipated annual DCQ reduction factor.

Once the DCQ has been reduced automatically below the specified initial value, the DCQ is multiplied by the anticipated reduction factor at the start of each contract period.

If you are using the accelerated iteration scheme (this is always the case for ECLIPSE 300 and controlled by item 2 of RUNSPEC section keyword GASFIELD for ECLIPSE 100), it is not necessary to have an anticipated reduction factor, as the scheme chooses a starting value based on the previous period's reduction in DCQ. This item should therefore be defaulted in general. If a value is specified here then it is used instead of the value calculated by ECLIPSE.

ECLIPSE 100

If you are not using the accelerated iteration scheme, an anticipated reduction factor could help reduce the number of iterations required to converge the DCQ by bringing the starting value nearer to the final value. For example, during the first pass of the contract period ECLIPSE calculates that the DCQ must be reduced to a value DCQ2. The second pass is solved with the DCQ at this value. If an anticipated reduction factor, FANT, has been set, ECLIPSE starts the first pass of the next contract period with an initial DCQ = DCQ2 × FANT.

RANGE: 0.01 to 1.0
DEFAULT: 1.0 (Giving no anticipated reduction)

See "Calculating the DCQ value" in the ECLIPSE Technical Description.

8. Maximum allowed number of iterations of the first pass of each contract period to calculate the DCQ.

This takes effect if you are using the accelerated iteration scheme (this is always the case for ECLIPSE 300 and controlled by item 2 of RUNSPEC section keyword GASFIELD for ECLIPSE 100) or if there is a limiting DCQ reduction factor defined in item 5. A warning will be issued if the DCQ has not converged within the maximum number of iterations.

ECLIPSE 100

If you are not using the accelerated iteration scheme, and there is no limiting DCQ reduction factor, the first pass is only performed once to calculate the DCQ. The resulting value of the DCQ will be pessimistic if there has been a significant reduction.

DEFAULT: 3
9. Convergence tolerance for the DCQ calculation when using the accelerated iteration scheme. 

The value must lie between 0.0 and 1.0. A value of 0.1, for example, accepts the DCQ as converged when its change in value over an iteration is 10% or less. The minimum of the two DCQ values is chosen, so that the DCQ is on the safe side of the true result. A warning is issued if the DCQ has not converged after the maximum number of iterations specified in item 7.

This item is ignored if you are not using the accelerated iteration scheme.

RANGE: 0.0 to 1.0

DEFAULT: 0.1

End the data with a slash (/)

--- CAUTION: If an Annual Scheduling file (ASF) is being used with contract periods of varying length some caution must be exercised. It is allowable, for example, to perform a 6 month contract period followed by 3 quarterly contract periods. The first two quarters take the ASF to the end of its year and it then starts again for the third quarter. However, it is not allowable to do 3 quarterly contract periods followed by a half year contract period; the ASF does not wrap around within a contract period. Essentially, the end of an ASF year must coincide with the end of a contract period. To achieve the second scenario it would be necessary to redefine the ASF starting at the appropriate month before the second use of GASP.ERIO.

--- Examples

To advance the simulation through three and a half years using quarterly contract periods, with the DCQ initially 100,000 but reduced when necessary to obey the swing requirement, and a new timestep starting on the first of each month. For ECLIPSE 100, the accelerated iteration scheme has been selected in RUNSPEC section keyword GASFIELD. A small limiting DCQ reduction factor is applied (0.1), and 4 iterations of the first pass are allowed for convergence to 5% accuracy.

**Example 1**

<table>
<thead>
<tr>
<th>GASPERIO</th>
<th>-- No of</th>
<th>Period</th>
<th>Initial</th>
<th>Swing</th>
<th>Monthly</th>
<th>Lim DCQ</th>
<th>Ant DCQ</th>
<th>Max no</th>
<th>Conv</th>
</tr>
</thead>
<tbody>
<tr>
<td>-- periods</td>
<td>Length</td>
<td>DCQ</td>
<td>req</td>
<td>t/steps</td>
<td>reduc</td>
<td>reduc</td>
<td>DCQ its</td>
<td>tol</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>3</td>
<td>100000</td>
<td>PER</td>
<td>1*</td>
<td>0.1</td>
<td>1*</td>
<td>4</td>
<td>0.05</td>
<td></td>
</tr>
</tbody>
</table>

**Example 2**

To continue the simulation for another year with the same DCQ iteration parameters but using two six month contract periods. The initial DCQ should be defaulted now.

<table>
<thead>
<tr>
<th>GASPERIO</th>
<th>-- No of</th>
<th>Period</th>
<th>Initial</th>
<th>Swing</th>
<th>Monthly</th>
<th>Lim DCQ</th>
<th>Ant DCQ</th>
<th>Max no</th>
<th>Conv</th>
</tr>
</thead>
<tbody>
<tr>
<td>-- periods</td>
<td>Length</td>
<td>DCQ</td>
<td>req</td>
<td>t/steps</td>
<td>reduc</td>
<td>reduc</td>
<td>DCQ its</td>
<td>tol</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>6</td>
<td>1*</td>
<td>PER</td>
<td>1*</td>
<td>0.1</td>
<td>1*</td>
<td>4</td>
<td>0.05</td>
<td></td>
</tr>
</tbody>
</table>

--- Keywords

865
**GASSATC**

**Initial saturated coal gas concentration**

The keyword should be followed by one real number for every matrix grid block, specifying the initial saturated gas concentration in the coal. The data field should be terminated by a slash (/).

UNITs: \( \text{sm}^3/\text{m}^3 \) (METRIC), Mscf/ft\(^3\) (FIELD), scc/cc (LAB)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword **DIMENS**. Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*). Data need only be supplied for the matrix cells, that is the upper half of the grid.

In ECLIPSE 100. This information is used to scale the Langmuir isotherm, as specified by the **LANGMUIR** keyword, at the initial pressure in the reservoir. This is an alternative to the **MLANG** keyword, which provides scaling of the Langmuir isotherm at the maximum fracture pressure entry in each of the **LANGMUIR** tables. Both GASSATC and MLANG are entirely optional. If neither of these keywords is present, the information provided in the **LANGMUIR** keyword is used without any scaling.

In ECLIPSE 300. This information is used to scale the Langmuir isotherm, as specified by the **LANGMEXT** or **LANGMUIR** keyword, at the initial pressure and composition in the reservoir for one chosen component. The component used is specified by the **GASCCMP** keyword. The same scaling factor is used for all the components. For the general case the keyword **GASADEC** can be used.

Note that the keyword cannot be used together with **LANGMULT**, **LANGMULC** or **GASADEC**.

Only values for coal cells are used. Note that the composition (YMF) and pressure value of the coal cell is used and not the connected pore volume (fracture) composition and pressure.

**Example**

With \( \text{NDIVIX}=8, \text{NDIVIY}=6, \text{NDIVIZ}=10 \) specified in RUNSPEC keyword **DIMENS**:

```
GASSATC
48*89.1 48*87.3 48*85.1 48*85.0 48*85.0 /
```

---

**Keywords**

866
**GASSOL**

**Specify number of gas components to dissolve in the aqueous phase**

This keyword turns on the general specification of compositional gas, for example a flue gas, for gas injection, allowing a specified number, \( NCMPHW \), of gas components to dissolve in the aqueous phase. This remains isothermal and there is no water in the gas phase. This keyword cannot be used if directional saturation table end-point scaling is used (switched on by setting the first item of keyword ENDSCALE to DIRECT).

This keyword is read in the RUNSPEC section to define the number of components allowed to dissolve in the aqueous phase in a compositional run. GASSOL contains one item, the number of components, \( NCMPHW \), to be allowed to dissolve in the aqueous phase. The total number of components in a compositional run is specified by keyword COMPS in the RUNSPEC section. If present it is read, and an error message is produced if the item’s value is less than zero or greater than specified with the COMPS keyword. There are several optional associated keywords:

- **SOLUAQA** Defines the solubility data.
- **RSWVD** This specifies the initial variation of the aqueous gas-water ratio versus depth for each component.
- **RSW** This specifies the initial variation of the aqueous gas-water ratio by enumeration for each component.
- **AMF** This specifies the initial composition of the aqueous gas system for each component.
- **VISCAQA** This specifies coefficients of Ezrokhi’s method for aqua viscosity for each component.
- **DENAQA** This specifies coefficients of Ezrokhi’s method for aqua density for each component.

For further information see "GASSOL solution in the aqueous phase" in the ECLIPSE Technical Description.

**Example**

Confirm that five components (\( NCMPHW=5 \)) are allowed to dissolve in the aqueous phase:

```
GASSOL
5 /
```
Gas viscosity function

This keyword can only be used with the THERMAL option.

For the $i^{th}$ oil component, the gas viscosity is fitted according to the equation:

$$\mu_g^i = A \cdot T^B$$

where $T$ is the temperature (K (METRIC), °R (FIELD), K (LAB), K (PVT-M)).

The gas phase viscosity is calculated as

$$\mu_g^{} = \sum_{i=1}^{N_c+1} \mu_g^i Y_i$$

where $Y_i$ are the component mole fractions in the gas phase.

The keyword is followed by two records, each containing $N_c$ columns of data, described below, followed by a blank record, containing only a slash.

1. **A coefficients**
   - A set of $N_c$ values is entered, one for each hydrocarbon component
   - UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

2. **B coefficients**
   - A set of $N_c$ values is entered, one for each hydrocarbon component

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as the 9th parameter of TABDIMS.

**Example**

```
GASVISCF
  -- SGAS HEAVY
  0.5E-4 1.0E-4
  0.9 0.9
/
```
GASVISCT  Gas viscosity versus temperature table

This keyword can only be used with the THERMAL option.

The data comprises NTPVT tables (the item 2 of keyword TABDIMS) of gas phase component viscosities versus temperature for each pressure table. Each table consists of $N_c + 1$ columns of data (described below), and is terminated by a slash (/).

1. Temperature values.
   The values should increase monotonically down the column.
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

2. The corresponding values of gas component viscosity.
   The values should increase or stay level down the column.
   UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

For temperature values outside the interval specified, constant value extrapolation is performed.

The gas phase viscosity is calculated as

$$
\mu_g = \sum_{i=1}^{N_c+1} \mu_{g_i} Y_i
$$

where $\mu_{g_i}$ are the gaseous component viscosities and $Y_i$ are the component mole fractions in the gas phase.

Example

```
<table>
<thead>
<tr>
<th>Temperature</th>
<th>Viscosity1</th>
<th>Viscosity2</th>
</tr>
</thead>
<tbody>
<tr>
<td>75</td>
<td>0.0143</td>
<td>0.0180</td>
</tr>
<tr>
<td>100</td>
<td>0.0149</td>
<td>0.0188</td>
</tr>
<tr>
<td>150</td>
<td>0.0161</td>
<td>0.0203</td>
</tr>
<tr>
<td>200</td>
<td>0.0172</td>
<td>0.0218</td>
</tr>
<tr>
<td>250</td>
<td>0.0184</td>
<td>0.0233</td>
</tr>
<tr>
<td>300</td>
<td>0.0196</td>
<td>0.0247</td>
</tr>
<tr>
<td>350</td>
<td>0.0207</td>
<td>0.0262</td>
</tr>
<tr>
<td>500</td>
<td>0.0241</td>
<td>0.0305</td>
</tr>
<tr>
<td>15000</td>
<td>0.2946</td>
<td>0.3723</td>
</tr>
</tbody>
</table>
/
```

Keywords

```
GASVISCT
```

869
Use water-gas model

The GASWAT option provides a method of modeling gas phase/aqueous phase equilibriums using an equation of state. The Peng Robinson equation of state is modified following the suggestions of Soreide and Whitson to obtain accurate gas solubilities in the aqueous phase.

This option is described in detail in "GASWAT option" in the ECLIPSE Technical Description.

The keyword GASWAT contains no items.
**GASYEAR**

**Advances simulator over ‘n’ gas field contract years**

The GASYEAR keyword can be used instead of **DATES**, **TIME** or **TSTEP** to advance the simulation over an integral number of contract years, when running the gas field operations model. For contract periods that are less than a year, keyword **GASPERIO** must be used instead. The GASYEAR keyword activates two features of the model:

- Generation of the annual production profile from the profile factors set in keyword **SWINGFAC** or **GSWINGF**.
- Automatic reduction of the DCQ when necessary to obey the swing factor requirement, using the swing factors set in keyword **SWINGFAC** or **GSWINGF**.

The **SWINGFAC** or **GSWINGF** keyword must have been entered previously to activate the Gas Field Operations Model.

The keyword should be followed by some or all of these data items, terminated with a slash (/):

1. **Number of contract years to simulate.**
   
The contract year begins at the current date when the GASYEAR keyword is read. An error will be flagged if this is not the first day of a month. At the end of each contract year, ECLIPSE writes an output report and dumps a restart file if required.

2. **The initial DCQ (Daily Contracted Quantity).**
   
   When there is a single gas supply contract on the FIELD (keyword **SWINGFAC**), the initial DCQ must be specified in the first entry of the GASYEAR keyword. It should be defaulted in subsequent entries, unless you wish to reset its value.

   When there are separate gas supply contracts on different groups (that is keyword **GSWINGF** has been used to activate the multiple contract group facility; see "Applying separate gas supply contracts to individual groups" in the ECLIPSE Technical Description), this item should be defaulted. Any value entered here is ignored. Instead, the initial DCQs of the contract groups must be set with keyword **GDCQ** before the GASYEAR keyword is read.

   **UNITS:** sm$^3$/day (METRIC), Mcf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

   **ECLIPSE 100 only**

   If item 1 of **DCQDEFN** is set to ‘ENERGY’:

   **UNITS:** kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB), kJ/day (PVT-M)

   **DEFAULT:** Current value of DCQ

3. **Swing requirement for automatic reduction of the DCQ**

   **YEAR** or **YES**  The field (or each contract group) must be able to produce at the DCQ multiplied by the monthly swing factor, for the whole contract year.

   **PRO**  The field (or contract group) must be able to raise its production rate from DCQ x profile to DCQ x swing for an instant at any time in the contract year.

   **ACQ**  The field (or contract group) must be able to produce at the DCQ multiplied by the monthly swing factor continuously until it has produced its annual contracted quantity (= 365.25 x DCQ).
The field (or contract group) must be able to produce at the DCQ multiplied by the monthly swing factor continuously, until it has produced a fraction of its annual contracted quantity proportional to the sum of profile factors up to the end of the specified month.

(JLY is an acceptable alternative to JUL.)

NO The swing requirement is not tested. The DCQ remains at its current value, even if the field (or contract group) cannot produce at its target rate.

items 5 - 8 of this keyword will be ignored

DEFAULT: YEAR

YEAR is a suitable option for cases where the monthly swing factors are the same as the profile factors, as it ensures the production profile fits underneath the delivery capacity decline curve. But for cases where the swing factors exceed the profile factors, the PRO or ACQ options may be more suitable. For more discussion of these options, see "Interpretations of the swing requirement" in the ECLIPSE Technical Description.

When using the multiple contract groups facility (initiated with keyword GSWINGF), only the options YEAR and NO are allowed if there is more than one variable-DCQ contract group (see keyword GDCQ). If all contract groups except one have fixed DCQs, all the above options are allowed.

4. Should the timesteps be limited so that each month starts with a new timestep? - YES or NO

(The word may be abbreviated, only the first character is significant)

YES ECLIPSE automatically limits each timestep so that it does not go beyond the end of the current month. This ensures that there is at least one Summary file entry per month, and that the network (if using the Network option) will be balanced at least once per month.

NO ECLIPSE only limits the timestep to the end of the month if the production rate target changes, or if there is an operation on the annual scheduling file. All other means of limiting the timestep (see keyword TUNING) still remain in force.

DEFAULT: YES

5. Limiting DCQ reduction factor, FLIM, allowed in a single iteration of the first pass of the contract year.

This restricts the amount by which the DCQ can be reduced in a single iteration of the first pass of the contract year. If the DCQ needs to be reduced to a fraction less than the value specified here, it is reduced only to this specified fraction (DCQ × FLIM) and the first pass of the contract year is solved again, subject to the upper limit in item 7 on the number of iterations.

If you are using the accelerated iteration scheme (this is always the case for ECLIPSE 300 and controlled by item 2 of RUNSPEC section keyword GASFIELD for ECLIPSE 100), it is not necessary to have a limiting reduction factor. Although a small value (for example 0.1) could be useful when the field or a contract group is getting close to the end of its life. This would stop the iteration immediately jumping to a DCQ of zero when it may still be able to operate with a small value of the DCQ.
If you are not using the accelerated iteration scheme, the DCQ reduction must be limited more strictly to obtain accurate results, to prevent it from substantially overshooting the correct value (it will not increase again). The nearer the reduction factor is to 1.0 the more accurate will be the result, but at the expense of requiring more iterations of the first pass to converge it.

**RANGE:** 0.0 to 0.99

**DEFAULT:** 0.0 (Allowing unlimited reduction)

See "Calculating the DCQ value" in the *ECLIPSE Technical Description*.

6. **Anticipated annual DCQ reduction factor.**

Once the DCQ has been reduced automatically below the specified initial value, the DCQ is multiplied by the anticipated reduction factor at the start of each contract year.

If you are using the accelerated iteration scheme (this is always the case for ECLIPSE 300 and controlled by item 2 of **RUNSPEC** section keyword `GASFIELD` for ECLIPSE 100), it is not necessary to have an anticipated reduction factor, as the scheme chooses a starting value based on the previous year's reduction in DCQ. This item should therefore be defaulted in general. If a value is specified here then it is used instead of the value calculated by ECLIPSE.

**RANGE:** 0.01 to 1.0

**DEFAULT:** 1.0 (Giving no anticipated reduction)

See "Calculating the DCQ value" in the *ECLIPSE Technical Description*.

7. **Maximum allowed number of iterations of the first pass of each contract year to calculate the DCQ.**

This takes effect if you are using the accelerated iteration scheme (this is always the case for ECLIPSE 300 and controlled by item 2 of **RUNSPEC** section keyword `GASFIELD` for ECLIPSE 100) or if there is a limiting DCQ reduction factor defined in item 5. A warning will be issued if the DCQ has not converged within the maximum number of iterations.

**DEFAULT:** 3

8. **Convergence tolerance for the calculation DCQ when using the accelerated iteration scheme.**

The value must lie between 0.0 and 1.0. A value of 0.1, for example, accepts the DCQ as converged when its change in value over an iteration is 10% or less. The minimum of the two DCQ values is chosen, so that the DCQ is on the safe side of the true result. A warning is issued if the DCQ has not converged after the maximum number of iterations specified in item 7.

**DEFAULT:** 0.1
End the data with a slash (/).

See also the GASPERSIO keyword.

**Examples**

**Example 1**
To advance the simulation through four years, with the DCQ initially 100,000 but reduced when necessary to obey the swing requirement, and a new timestep starting on the first of each month. For ECLIPSE 100, the accelerated iteration scheme has been selected in RUNSPEC section keyword GASFIELD. A small limiting DCQ reduction factor is applied (0.1), and 4 iterations of the first pass are allowed for convergence to 5% accuracy.

<table>
<thead>
<tr>
<th>GASYEAR</th>
<th>-- No of years</th>
<th>Initial DCQ</th>
<th>Swing req</th>
<th>Monthly t/steps</th>
<th>Lim DCQ reduc</th>
<th>Ant DCQ reduc</th>
<th>Max no DCQ its</th>
<th>Conv tol</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>100000</td>
<td>YEAR</td>
<td>1*</td>
<td>0.1</td>
<td>1*</td>
<td>4</td>
<td>0.05</td>
<td>/</td>
</tr>
</tbody>
</table>

**Example 2**
To continue the simulation for another two years with the same DCQ iteration parameters. The initial DCQ should be defaulted now.

<table>
<thead>
<tr>
<th>GASYEAR</th>
<th>-- No of years</th>
<th>Initial DCQ</th>
<th>Swing req</th>
<th>Monthly t/steps</th>
<th>Lim DCQ reduc</th>
<th>Ant DCQ reduc</th>
<th>Max no DCQ its</th>
<th>Conv tol</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>1*</td>
<td>YEAR</td>
<td>1*</td>
<td>0.1</td>
<td>1*</td>
<td>4</td>
<td>0.05</td>
<td>/</td>
</tr>
</tbody>
</table>

**Example 3**
To advance the simulation through one year with the DCQ fixed at 50000, and timesteps limited just by rate changes.

<table>
<thead>
<tr>
<th>GASYEAR</th>
<th>-- No of years</th>
<th>Initial DCQ</th>
<th>Swing req</th>
<th>Monthly t/steps</th>
<th>Lim DCQ reduc</th>
<th>Ant DCQ reduc</th>
<th>Max no DCQ its</th>
<th>Conv tol</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>50000</td>
<td>NO</td>
<td>NO</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>/</td>
</tr>
</tbody>
</table>
GCALECON  Energy and calorific value economic limit data for groups

The keyword allows minimum economic limits to be set for the energy rate and/or the mean calorific value of individual groups and the field, when running the gas calorific value control option (see “Gas calorific value control” in the ECLIPSE Technical Description).

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root,
   or FIELD (for field economic limit data)
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Minimum energy production rate
   UNITS: kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB), kJ/day (PVT-M)
   DEFAULT: 0.0
   If the group’s energy production rate falls below the specified minimum, all the producers in the group are shut or stopped (see item 9 in keyword WELSPECS). A zero or negative value switches off this limit.

3. Minimum calorific value
   UNITS: kJ/sm³ (METRIC), Btu/Mscf (FIELD), J/scc (LAB), kJ/sm³ (PVT-M)
   DEFAULT: 0.0
   If the group’s mean calorific value falls below the specified minimum, all the producers in the group are shut or stopped (see item 9 in keyword WELSPECS). A zero or negative value switches off this limit.

4. End run flag
   YES  The run stops at the next report time if all the producers in the group are shut or stopped for any reason, provided the group has had at least one producer open previously
   NO   The run continues regardless
   DEFAULT: NO

Note: If the GECON keyword is entered subsequently to GCALECON, item 8 of the GECON keyword overwrites the end run flag defined here.

End each data record with a slash (/).
End the set of records with a blank record, containing just a slash.
Example

GCALECON
   G1  1.0E7  28000  YES/
   /
Calorific value controls for groups

The **GCONCAL** keyword sets a target mean calorific value for the gas produced by a group (or the field), when running the gas calorific value control option. The group’s gas production rate must simultaneously be under guide rate group control (keyword **GCONPROD**). Mean calorific value targets are met by adjusting the guide rates to obtain a suitable mix of gases of different calorific values. See "Adjusting guide rates to control mean calorific value" in the ECLIPSE Technical Description for further information.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root,
   or **FIELD** (for field calorific value controls)
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.
2. Target mean calorific value of the produced gas
   UNITS: kJ/sm^3 (METRIC), Btu/Mscf (FIELD), J/scc (LAB), kJ/sm^3 (PVT-M)
   DEFAULT: Infinity (1.0E20), which will turn off the calorific value control
3. Action if the calorific value target cannot be met after all available production rule actions have been tried (because there are no wells under group control whose calorific values straddle the target)
   **NONE** Take no further action; allow the mean calorific value to drift.
   **RATE** Cut the group’s gas rate target successively, by the rate reduction factor in item 4, until wells with calorific values straddling the target are brought back under group control.
   See "Action when calorific value target cannot be met" in the ECLIPSE Technical Description for further details.
   DEFAULT: NONE
4. Rate reduction factor for the **RATE** action in item 3
   DEFAULT: 0.9

End each data record with a slash (/).

End the set of records with a blank record, containing just a slash.

**Example**

```
GCONCAL
FIELD  35000.0  RATE  0.95 /
```
The **GCONENG** keyword sets a target or limiting energy production rate for groups (or the field), when running the gas calorific value control option. A group’s energy production target is apportioned between its subordinate wells according to their energy guide rates. See "Controlling the energy production rate" in the *ECLIPSE Technical Description* for further information.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root,  
   or **FIELD** (for field energy production rate control)  
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Energy production rate target or upper limit  
   **UNIT**: kJ/day (**METRIC**), Btu/day (**FIELD**), J/hr (**LAB**), kJ/day (**PVT-M**)  
   **DEFAULT**: Infinity (1.0E20)  

End each data record with a slash (/).

End the set of records with a blank record, containing just a slash.

**Example**

```plaintext
GCONENG
FIELD  1.0E9 /
```
Injection rate controls/limits for groups/field

This keyword specifies the injection targets and limits for groups.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated at any item after item 1. The remaining items assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root
   or FIELD (for field rate control data)

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Phase to which the following controls/limits apply

   WATER  Water injection controls.

   GAS  Gas injection controls.

   GAS

   When running in compositional mode, the nature of the injected gas must also be specified with keyword GINJGAS, unless the group is under control of a higher level group with gas injection controls where the injection fluid is defined.

   ECLIPSE 300 only

   OI  Oil injection controls.

   Injection controls and limits can be applied to one or more phases in any group. A separate data record is needed for each phase.

3. Injection rate control mode

   NONE  No immediate control of injection rate.

   RATE  The group/field surface injection rate of the phase in item 2 is controlled to meet the target specified in item 4.

   RESV  The group/field reservoir volume injection rate of the phase in item 2 will be controlled so that the total reservoir volume injection rate of the group/field meets the target specified in item 5.

   REIN  The group/field surface injection rate of the phase in item 2 is controlled to equal the group/field production rate of the phase times the reinjection fraction specified in item 6.

   VREP  The group/field reservoir volume injection rate of the phase in item 2 is controlled so that the total reservoir volume injection rate of the phase in item 2 is controlled to meet the target specified in item 13.

   ECLIPSE 300 only

   WGRA  The group/field wet gas injection rate of the phase in item 2 is controlled to equal the total reservoir volume injection rate of the phase in item 2 times the voidage replacement fraction specified in item 7.

Keywords

ECLIPSE Reference Manual

879
The group is immediately under control from a higher level group or the field, injecting its share of the higher group’s or field’s target rate, according to its guide rate (which must be set in items 9 and 10 in this keyword).

DEFAULT: NONE

4. Surface injection rate target or upper limit for the phase in item 2. User defined arguments can be set with keywords UDQ and UDADIMS.

UNITS: \( \text{sm}^3/\text{day} \) (METRIC), \( \text{stb/day} \) for oil or water (FIELD), \( \text{Mscf/day} \) for gas (FIELD), \( \text{scc/hr} \) (LAB), \( \text{sm}^3/\text{day} \) (PVT-M)

DEFAULT: No target or limit

5. Total reservoir volume injection rate target or upper limit. User defined arguments can be set with keywords UDQ and UDADIMS.

If a non-default value is specified here, the phase in item 2 is declared the top-up phase. Its target or limiting reservoir volume injection rate will be equal to the value specified here minus the reservoir volume injection rate of the other phases. The phase injection rate is calculated to top up the total group or field injection to the required reservoir volume rate, after allowing for any injection of the other phases. There can be only one top-up phase at any given time in the simulation run.

UNITS: \( \text{rm}^3/\text{day} \) (METRIC), \( \text{rb/day} \) (FIELD), \( \text{rcc/hr} \) (LAB), \( \text{rm}^3/\text{day} \) (PVT-M)

DEFAULT: No target or limit

6. Reinjection fraction target or upper limit for the phase in item 2. User defined arguments can be set with keywords UDQ and UDADIMS.

The reinjection fraction is applied to the group or field production rate of the phase in item 2.

**ECLIPSE 100**

For gas, the consumption rate of the group and any subordinate groups is subtracted from the production rate, and the gas import rate is added to it, before multiplying by the reinjection fraction (see keyword GCONSUMP).

**ECLIPSE 300**

For gas, the reinjection fraction is applied to the group’s available gas. The available produced gas rate of a group is by default equal to the group’s gas production rate minus its fuel rate (keyword GRUPPFUEL) minus its sales target (keyword GRUPSALE); however, this definition can be altered using keyword WTAKEGAS. Any advance gas allocated to the group (keyword GADVANCE) is also added to the available gas. This limit ignores makeup gas (keyword GINJGAS, item 4); it should not be applied if makeup gas is available, which is treated as an infinite resource. In the case of a reinjection fluid defined by a stream or mixture, the specified fraction of the available surface gas production rate is applied as a wet gas reinjection rate target.

The reinjection fraction can be applied to the production rate of a different group by entering the required group’s name in item 11.

**ECLIPSE 100**

DEFAULT: No target or limit

**ECLIPSE 300**

DEFAULT: 1 (if the injection rate control mode is REIN), otherwise no target or limit.

7. Total voidage replacement fraction target or upper limit. User defined arguments can be set with keywords UDQ and UDADIMS.

If a non-default value is specified here, the phase in item 2 is declared the ‘top-up’ phase. Its target or limiting reservoir volume injection rate will be equal to the value specified here times the group/field
production voidage rate, minus the reservoir volume injection rate of the other phases. The phase injection rate is calculated to ‘top up’ the total group or field injection to the required voidage replacement fraction, after allowing for any injection of the other phases. There can be only one ‘top-up’ phase at any given time in the simulation run.

The voidage replacement fraction can be applied to the reservoir volume production rate of a different group by entering the required group’s name in item 12.

**ECLIPSE 100**

DEFAULT: No target or limit

**ECLIPSE 300**

DEFAULT: 1 (if the injection rate control mode is VREP), otherwise no target or limit.

8. Is the group free to respond to a higher level injection rate target?

**YES** Injection from the group is cut back as necessary to meet any rate target/limit imposed at a higher level

**NO** The group injects at its own capacity or target irrespective of any higher level rate target or limit

This item is ignored if the data record applies to the FIELD.

DEFAULT: **YES**

9. Group’s injection guide rate for the phase in item 2, a dimensionless number governing the group’s share of a higher level injection target

A group needs a guide rate only if it is required to inject a specified proportion of a higher level target. The group is then placed under rate control, with a target rate set in proportion to its guide rate. The definition of this guide rate is specified in item 10. It can either be set to a fixed surface rate or reservoir volume rate, or adjusted each timestep to reflect the group’s voidage replacement requirements.

If no guide rate is specified, the group's share of a higher level injection target is governed by the guide rates of its subordinate injection wells, or any subordinate groups with injection guide rates. (The well guide rates are set by default equal to the wells’ injection potentials, but can be specified directly by using the keyword WGRUPCON). The group will essentially be transparent to higher level injection targets, which will be apportioned directly down to its subordinate wells or groups with guide rates.

Set this item to a positive value if you wish to give the group a fixed injection guide rate for this phase (with RATE or RESV set in item 10).

Default this item if the group is to have no injection guide rate for this phase, or if the guide rate definition in item 10 is VOID or NETV.

**ECLIPSE 300**

If a zero guide rate is specified, the group’s guide rate is set at the beginning of each timestep to be equal to the group’s injection potential - that is, the sum of the injection potentials of all of its subordinate open injectors.

This item is ignored if the data record applies to the FIELD.

DEFAULT: No fixed guide rate


**RATE** The guide rate applies to group’s surface injection rate (so the surface injection rate target will be in proportion to the guide rate).
RESV The guide rate applies to group’s reservoir volume injection rate (so the reservoir volume injection target is in proportion to the guide rate).

**ECLIPSE 100 only** VOID The guide rate is set at the beginning of each timestep equal to the group’s voidage production rate. This setting can be used to apportion an injection target between groups in proportion to their voidage production rates. The entry in item 9 is ignored.

**ECLIPSE 100 only** NETV The guide rate is set at the beginning of each timestep equal to the group’s net voidage rate (that is its voidage production rate minus the reservoir volume injection rate of any other phases). This setting can be used to apportion an injection target between groups in proportion to their top-up needs for voidage replacement. It can only be applied if the phase in item 2 is the ‘top-up’ phase. The entry in item 9 is ignored.

' The group has no injection guide rate for this phase.

Higher level injection targets is apportioned directly down to its subordinate wells, or any subordinate groups with injection guide rates.

The item is ignored if the data record applies to the FIELD.

DEFAULT: ' (The group has no injection guide rate.)

11. Name of the group whose production rate the reinjection fraction in item 6 should be applied to.

The group in item 1 can be made to reinject a fraction of another group’s production rate, by entering the name of this other group here.

DEFAULT: The group in item 1 reinjects a fraction of its own production rate

12. Name of the group whose reservoir volume production rate the voidage replacement fraction in item 7 should be applied to.

The group in item 1 can be made to replace a fraction of another group’s voidage, by entering the name of this other group here.

DEFAULT: The group in item 1 replaces a fraction of its own voidage

**ECLIPSE 300 only** 13. Wet gas injection rate target or upper limit for the phase in item 2 (this should only be set if the phase in item 2 is GAS)

Wet gas rate is related to the hydrocarbon moles injected using a unity Z-factor at surface conditions.

UNITs: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: No target or limit

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

**Note:** Setting non-default values in item 5 or 7, or entering NETV in item 10, causes the phase in item 2 to become the top-up phase. Its injection target depends not only on the production flows but also on the injection flows of any other injected phases. Thus the top-up phase flows must be calculated after all other injection phases have been processed. There can only be one ‘top-up’ phase in the simulation at any one time. Two groups cannot have different ‘top-up’ phases. However, the top-up phase can be changed during the run. If for example you initially inject water to replace the voidage, but wish to change to voidage
replacement with gas later in the run, it is first necessary to remove all top-up phase controls and limits from the water phase. Thus GCONINJE must first be used for the water phase to reset items 5 and 7 to their default values, and item 10 must not be set to NETV, before specifying voidage replacement controls for the gas phase. In this situation, the simplest way to remove the limits from the water phase is to change the injection rate control mode for the water phase to NONE.

**Examples**

![Five level hierarchy diagram](image)

*Figure 3.6. A five level hierarchy*

**Example 1**

The field reinjects all of its available gas, to be shared between the injection wells in proportion to their guide rates or potentials, subject to their individual rate and pressure limits.

```
GCONINJE
  FIELD GAS REIN 2* 1.0 /
```

**Example 2**

The field injects 30,000 Mscf/day of gas, subject to PLAT-A and PLAT-B not injecting more than their available gas. SAT-B has a gas injection limit of 10,000 Mscf/day.

```
GCONINJE
  FIELD GAS RATE 30000 /
  'PLAT*' GAS NONE 2* 1.0 /
  SAT-B GAS NONE 10000 /
```

**Example 3**

As Example 2, but the injected gas is to be distributed between each bottom-level group in proportion to its voidage production (subject to the platform constraints and the group’s injection potentials). Since the bottom-level groups are given guide rates, they are placed under rate control from a higher level.

```
GCONINJE
  FIELD GAS RATE 30000 /
  'PLAT*' GAS NONE 2* 1.0 /
  SAT-B GAS NONE 10000 /
  'GR-*' GAS FLD 6* VOID /
```

**ECLIPSE 100**

Keywords 883
Example 4

The field injects water to replace half its production voidage. The water injection is shared between the injection wells in proportion to their guide rates or potentials, subject to their individual rate and pressure limits.

```
GCONINJE
  FIELD WAT VREP 3* 0.5 /
/
```

Example 5

As Example 1, but with the remainder of the field’s voidage replacement needs being met by water injection.

```
GCONINJE
  FIELD GAS REIN 2* 1.0 /
  FIELD WAT VREP 3* 1.0 /
/
```

Example 6

As Example 3, but with the remainder of the field’s voidage replacement needs being met by water injection, subject to a maximum rate of 6000 stb/day on PLAT-A and PLAT-B. The water injection is to be distributed, whenever possible, between the bottom-level groups in proportion to their net voidage rates after allowing for gas injection.

```
GCONINJE
  FIELD GAS RATE 30000 /
  'PLAT*' GAS NONE 2* 1.0 /
  SAT-B GAS NONE 10000 /
  'GR-*' GAS FLD 6* VOID /
  FIELD WAT VREP 3* 1.0 /
  'PLAT*' WAT NONE 6000 /
  'GR-*' WAT FLD 6* NETV /
/
```
Group/field production limits for prioritization

This keyword sets the production rate limits for groups using the prioritization option. The option must be activated with the keyword PRIORITY. See the PRIORITY keyword for a description of this option, and how it can be mixed with the guide rate method of group control.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

Note: When switching between group prioritization and group production rate control methods WELOPEN can be used to open wells temporarily closed under prioritization control.

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root
   or FIELD (for field rate limits)
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Oil production rate upper limit. User defined arguments can be set with keywords UDQ and UDADIMS.
   UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)
   DEFAULT: No limit

3. Procedure on exceeding oil rate limit
   NONE  Do nothing
   CON   Shut worst-offending connection in worst-offending well
   +CON  Shut worst-offending connection and all below it in worst-offending well (see keyword COMPORD)
   WELL  Shut or stop worst-offending well
          (item 9 in keyword WELSPECS)

   ECLIPSE 100 only
   PLUG Plug back worst-offending well (see keyword WPLUG)

   ECLIPSE 100 only
   PRI  Control group/field production rate by prioritization (using the first priority formula if two have been defined).

   ECLIPSE 100 only
   PR2 Control group/field production rate by prioritization using the second priority formula.
   (In ECLIPSE 300, CON+ is interpreted as +CON for back-compatibility)
   DEFAULT: NONE

   ECLIPSE 300
   Wells and connections are closed or plugged back at the end of the timestep in which the limit is exceeded (but see keyword WLIMTOL). The worst-offending well or connection here is the one producing the highest ratio of the violating phase to the well’s preferred phase (item 6 of keyword WELSPECS). See also keywords COMPLUMP and WORKLIM.
If a preferred phase is not defined, the worst offending well or connection here is the one producing the highest rate of the violating phase.

4. Water production rate upper limit. User defined arguments can be set with keywords UDQ and UDADIMS.

   UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scf/hr (LAB), sm$^3$/day (PVT-M)
   DEFAULT: No limit

5. Procedure on exceeding the water rate limit in item 4
   The choices are as defined in item 3.
   DEFAULT: NONE

6. Gas production rate upper limit. User defined arguments can be set with keywords UDQ and UDADIMS.

   UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scf/hr (LAB), sm$^3$/day (PVT-M)
   DEFAULT: No limit

7. Procedure on exceeding the gas rate limit in item 6
   The choices are as defined in item 3.
   DEFAULT: NONE

8. Liquid production rate upper limit. User defined arguments can be set with keywords UDQ and UDADIMS.

   UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scf/hr (LAB), sm$^3$/day (PVT-M)
   DEFAULT: No limit

9. Procedure on exceeding the liquid rate limit in item 8
   The choices are as defined in item 3.
   DEFAULT: NONE

10. Reservoir fluid volume production rate upper limit. User defined arguments can be set with keywords UDQ and UDADIMS.
    The procedure on exceeding this limit is PRI.

    UNITS: rm$^3$/day (METRIC), rb/day (FIELD), rcc/hr (LAB), rm$^3$/day (PVT-M)
    DEFAULT: No limit

11. Reservoir volume production balancing fraction upper limit. User defined arguments can be set with keywords UDQ and UDADIMS.
    This limits the group’s reservoir fluid volume production rate to the specified fraction (or multiple) of the group’s reservoir fluid volume injection rate.
    The procedure on exceeding this limit is PRI.
    DEFAULT: No limit

12. ECLIPSE 300 only: Wet gas production rate upper limit. User defined arguments can be set with keywords UDQ and UDADIMS.
Wet gas rate is related to the hydrocarbon moles produced using a unity Z-factor at surface conditions.

UNITS: $\text{m}^3/\text{day (METRIC)}$, $\text{Mscf/day (FIELD)}$, $\text{scf/hr (LAB)}$, $\text{m}^3/\text{day (PVT-M)}$

DEFAULT: No limit

13. ECLIPSE 300 only: Procedure on exceeding the wet gas rate limit in item 12

The choices are as defined in item 3.

DEFAULT: NONE

14. ECLIPSE 300 only: Surface gas volume production balancing fraction upper limit. User defined arguments can be set with keywords UDQ and UDADIMS.

This limits the group’s surface gas volume production rate so that the available gas (production gas minus fuel gas minus sales gas) is the specified fraction (or multiple) of the group’s surface gas volume injection rate. (Setting item 104 in OPTIONS3 to a non-zero value modifies the balancing so that it is the surface gas volume production which is limited to the specified fraction (or multiple) of the group’s surface gas volume injection rate.)

The procedure on exceeding this limit is PRI.

DEFAULT: No limit

15. ECLIPSE 300 only: Surface water volume production balancing fraction upper limit. User defined arguments can be set with keywords UDQ and UDADIMS.

This limits the group’s surface water volume production rate to the specified fraction (or multiple) of the group’s surface water volume injection rate.

The procedure on exceeding this limit is PRI.

DEFAULT: No limit

16. Linearly combined rate target or upper limit. Linear combination coefficients must be set with keyword LINCOM. User defined arguments can be set with keywords UDQ and UDADIMS.

UNITS: determined by keyword LCUNIT

DEFAULT: No target or limit

17. Procedure on exceeding the linearly combined rate limit in item 16

The choices are as defined in item 3.

DEFAULT: NONE

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also keywords GRUPTARG and PRIORITY.

Notes

For temperature values outside the interval specified, constant value extrapolation is performed. If a non-default limit is specified for any of the production balancing targets/limits (that is, any of items 11, 14 and 15), ECLIPSE will solve all the injectors before the producers at each timestep, instead of the other way round. While any group has a production balancing fraction upper limit, the following controls are not allowed because these facilities require the producers to be solved before the injectors.
• Group reinjection or voidage replacement targets/limits
• Well reinjection or voidage replacement targets/limits

**ECLIPSE 100 only**
• Guide rate definitions VOID or NETV in keyword `GCONINJE` item 10

**ECLIPSE 100 only**
• Sales gas control (keyword `GCONSALE`), which uses reinjection.

**ECLIPSE 300 only**
• Advanced or Make-up gas (keywords `GADVANCE` or `GINJGAS`).

**ECLIPSE 300 only**
• Injection limited by availability (keyword `WAVAILIM`).

**ECLIPSE 300**

If the composition of the injected gas stream is defined by the produced vapor from a group or well (keywords `GINJGAS` or `WINJGAS`), the composition will be taken from the previous converged Newton iteration.

**Example**

The field has an oil production rate limit of 10,000 stb/day. All groups with a name beginning with PLAT have liquid production rate limits of 6,000 stb/day. Wells will be brought on in decreasing order of priority until these limits are reached. The platforms also have gas rate limits of 20,000 Mscf/day. If these limits are exceeded the worst-offending wells will be worked over.

```
GCONPRI
FIELD 10000 PRI /
'PLAT*' 4* 20000 CON 6000 PRI /
/ 888
```
GCONPROD  Production rate controls/limits for groups or field

This keyword sets the production rate targets and limits for groups using the guide rate method of group control.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated at any item after item 1. The remaining items assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

Note: When switching between group prioritization and group production rate control methods WELOPEN can be used to open wells temporarily closed under prioritization control.

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root
   or FIELD (for field rate limits)

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several
groups in one record.

2. Production rate control mode

   NONE  No immediate control of production rate

   ORAT  Group/field oil production rate is controlled to meet the target specified in item 3

   WRAT  Group/field water production rate is controlled to meet the target specified in item 4

   GRAT  Group/field gas production rate is controlled to meet the target specified in item 5

   LRAT  Group/field liquid production rate is controlled to meet the target specified in item 6

   CRAT  Group/field linearly combined production rate is controlled to meet the target specified in
         item 20

   RESV  Group/field reservoir fluid volume production rate is controlled to meet the target specified
         in item 14

   PRBL  Group/field reservoir fluid volume production rate is controlled to meet the target production
         balancing fraction specified in item 15

   WGRA  Group/field wet gas production rate is controlled to meet the target specified in item 16

   CVAL  Group/field calorific rate is controlled to meet the target specified in item 17. Molar calorific
         values must be specified with the CALVAL keyword before this control mode may be used.

   PBGS  Group/field surface gas production rate is controlled to meet the target specified in item 18

   PBWS  Group/field surface water production rate is controlled to meet the target specified in item 19

Keywords

889
Group is immediately under control from a higher level group or the field, producing its share of the higher group’s or field’s target rate, according to its guide rate (which must be set in items 9 and 10 in this keyword).

**DEFAULT:** NONE

3. Oil production rate target or upper limit. User defined arguments can be set with keywords **UDQ** and **UDADIMS**.

**UNITS:** sm$^3$/day (**METRIC**), stb/day (**FIELD**), scc/hr (**LAB**), sm$^3$/day (**PVT-M**)

**DEFAULT:** No target or limit

4. Water production rate target or upper limit. User defined arguments can be set with keywords **UDQ** and **UDADIMS**.

**UNITS:** sm$^3$/day (**METRIC**), stb/day (**FIELD**), scc/hr (**LAB**), sm$^3$/day (**PVT-M**)

**DEFAULT:** No target or limit

5. Gas production rate target or upper limit. User defined arguments can be set with keywords **UDQ** and **UDADIMS**.

**UNITS:** sm$^3$/day (**METRIC**), Mscf/day (**FIELD**), scc/hr (**LAB**), sm$^3$/day (**PVT-M**)

**DEFAULT:** No target or limit

6. Liquid production rate target or upper limit. User defined arguments can be set with keywords **UDQ** and **UDADIMS**.

**UNITS:** sm$^3$/day (**METRIC**), stb/day (**FIELD**), scc/hr (**LAB**), sm$^3$/day (**PVT-M**)

**DEFAULT:** No target or limit

7. Procedure on exceeding a maximum rate limit

   **NONE**  Do nothing

   **CON**   Shut worst-offending connection in worst-offending well

   **+CON**  Shut worst-offending connection and all below it in worst-offending well (see keyword **COMPORD**)

   **WELL**  Shut or stop worst-offending well (item 9 in keyword **WELSPECS**)

   **ECLIPSE 100 only**

   **PLUG**  Plug back worst-offending well (see keyword **WPLUG**)

   **RATE**  Control group/field production rate to equal the violated upper limit

   (In ECLIPSE 300, **CON+** is interpreted as **+CON** for back-compatibility)

   **DEFAULT:** NONE

Wells and connections are closed or plugged back at the end of the timestep in which the limit is exceeded (but see keyword **WLIMITOL**). The ‘worst-offending’ well or connection here is the one producing the highest ratio of the violating phase to the well’s preferred phase (item 6 of keyword **WELSPECS**). See also **COMPLUMP** and **WORKLIM** for ECLIPSE 100.
If a preferred phase is not defined, the ‘worst offending’ well or connection here is the one producing the highest rate of the violating phase.

The procedure specified here applies to the four rate limits set in items 3 - 6, except for a rate limit that is a control target as specified in item 2. For control targets, the procedure is always RATE. The procedure can be reset for individual rate limits in items 11 - 13.

8. Is the group free to respond to a higher level production rate target?

YES production from the group is cut back as necessary to meet any rate target imposed at a higher level

NO the group produces at its own capacity or target irrespective of any higher level rate target

This item is ignored if the control data record applies to the FIELD.

DEFAULT: YES

9. Group’s production guide rate

A dimensionless number governing the group’s share of a higher level production rate target

A group needs a guide rate only if it is required to produce a specified proportion of a higher level rate target. The group is then placed under rate control, with a target rate set in proportion to its guide rate. The definition of this guide rate is specified in item 10. It can either be set for a specified phase, or calculated from the group's production potentials or (in ECLIPSE 100) its injection rate. If the guide rate phase in item 10 differs from the phase under control, the guide rate is translated into a guide rate for the controlled phase using the group’s production ratios at the beginning of each timestep.

If no guide rate is specified, the group’s share of a higher level rate target is governed by the guide rates of its subordinate wells, or any subordinate groups with guide rates. (The well guide rates are set by default equal to the wells’ production potentials, but can be specified directly by using the keyword WGRUPCON). The group is essentially ‘transparent’ to higher level production rate targets, which will be apportioned directly down to its subordinate wells or groups with guide rates.

Set this item to a positive value if you wish to give the group a fixed guide rate for a particular phase (OIL, WAT, GAS, LIQ, RES, WGA or CVAL) defined in item 10.

Default this item if the group is to have no guide rate, or if the guide rate definition in item 10 is INJV, POTN, or FORM.

If a zero guide rate is specified, this has the same effect as setting item 10 to POTN. (This option of setting the guide rate is only included for back-compatibility with old datasets. It should not be encouraged for new datasets.)

This item is ignored if the data record applies to the FIELD.

10. Definition of the group's guide rate in item 9.

OIL The group’s guide rate applies to the oil phase.

WAT The group’s guide rate applies to the water phase.

GAS The group’s guide rate applies to the gas phase.

LIQ The group’s guide rate applies to the liquid phase.

COMB The group’s guide rate applies to the linearly combined phase.
**WGA**  The group’s guide rate applies to the wet gas production.

**CVAL**  The group’s guide rate applies to the calorific value production.

**INJV**  The group’s guide rate is set at the beginning of each timestep equal to the group’s reservoir volume injection rate. This can be useful when the field has a production balancing target that is apportioned among the groups in proportion to their injection rates. See item 15. The entry in item 9 is ignored.

**POTN**  The group’s guide rate is set at the beginning of each timestep equal to the group’s production potential - which is the sum of the production potentials of all its subordinate open producers; see "Well potentials" in the ECLIPSE Technical Description. The entry in item 9 is ignored. (This option is primarily intended for use with master groups in the ECLIPSE 100 Reservoir Coupling facility. It is not recommended for general use outside this facility, as similar results can be obtained more efficiently by defaulting the well and group guide rates.)

**FORM**  The group's guide rate is calculated at specified intervals from a formula involving its production potentials, defined with the keyword `GUIDERAT`. This option can be used to decrease the contribution from groups with high GOR or water cut. The entry in item 9 will be ignored. (As with the POTN option, the FORM option is primarily intended for use with master groups in the ECLIPSE 100 Reservoir Coupling facility. Outside this facility, similar results may be obtained more efficiently by applying the guide rate formula directly to the wells.)

' '  The group has no production guide rate. Higher level production targets will be apportioned directly down to its subordinate wells, or any subordinate groups with guide rates.

The item will be ignored if the data record applies to the FIELD.

**DEFAULT:** ' ' (The group has no production guide rate)

11. Procedure on exceeding the water rate limit in item 4.
   The choices are as defined in item 7.
   If this item is defaulted, the procedure specified in item 7 is applied.

12. Procedure on exceeding the gas rate limit in item 5.
   The choices are as defined in item 7.
   If this item is defaulted, the procedure specified in item 7 is applied.

13. Procedure on exceeding the liquid rate limit in item 6.
   The choices are as defined in item 7.
   If this item is defaulted, the procedure specified in item 7 is applied.

14. Reservoir fluid volume production rate target or upper limit.
   The procedure on exceeding this limit is always RATE.
   **UNITS:** rm$^3$/day (METRIC), rb/day (FIELD), rcc/hr (LAB), rm$^3$/day (PVT-M)
   **DEFAULT:** No target or limit
15. Reservoir volume production balancing fraction target or upper limit.

This limits the group’s reservoir fluid volume production rate to the specified fraction (or multiple) of the group’s reservoir fluid volume injection rate.

DEFAULT: No target or limit

The procedure on exceeding this limit is always RATE.

**ECLIPSE 300 only**

16. Wet gas production rate target or upper limit

Wet gas rate is related to the hydrocarbon moles produced using a unity Z-factor at surface conditions.

UNITS: \( \text{sm}^3/\text{day (METRIC)}, \text{Mscf/day (FIELD)}, \text{scc/hr (LAB)}, \text{sm}^3/\text{day (PVT-M)} \)

DEFAULT: No target or limit

The procedure on exceeding this limit is always RATE.

**ECLIPSE 300 only**

17. Calorific rate target or upper limit

UNITS: \( \text{kJ/day (METRIC)}, \text{Btu/day (FIELD)}, \text{J/hr (LAB)}, \text{kJ/day (PVT-M)} \)

DEFAULT: No target or limit

The procedure on exceeding this limit is always RATE.

**ECLIPSE 300 only**

18. Surface gas production balancing fraction target or upper limit.

This limits the group’s surface gas volume production rate so that the available gas (production gas minus fuel gas minus sales gas) is the specified fraction (or multiple) of the group’s surface gas volume injection rate. (Setting OPTIONS3 item 104 to a non-zero value modifies the balancing so that it is the surface gas volume production which is limited to the specified fraction (or multiple) of the group’s surface gas volume injection rate.)

DEFAULT: No target or limit

The procedure on exceeding this limit is always RATE.

**ECLIPSE 300 only**

19. Surface water production balancing fraction target or upper limit

This limits the group’s surface water volume production rate to the specified fraction (or multiple) of the group’s surface water volume injection rate.

DEFAULT: No target or limit

The procedure on exceeding this limit is always RATE.

20. Linearly combined rate target or upper limit. Linear combination coefficients must be set with keyword LINCOM. User defined arguments can be set with keywords UDQ and UDADIMS.

UNITS: determined by keyword LCUNIT

DEFAULT: No target or limit

21. Procedure on exceeding the linearly combined rate limit in item 20.

The choices are as defined in item 7.

If this item is defaulted, the procedure specified in item 7 is applied.

End each data record with a slash (/).

End the set of records with a blank record, containing just a slash.
See also keywords GRUPTARG and WGRUPCON, and also GUIDERAT and PRORDER in ECLIPSE 100.

Notes

If a non-default value is specified for any of the production balancing targets/limits (that is, any of items 15, 18 and 19), or if in ECLIPSE 100 an INJV guide rate is specified in item 10, ECLIPSE solves all the injectors are solved before the producers at each timestep, instead of the other way round. While any group has a production balancing fraction target or limit, the following controls are not allowed:

- Group reinjection or voidage replacement targets/limits
- Well reinjection or voidage replacement targets/limits

**ECLIPSE 100 only**
- Guide rate definitions (VOID or NETV in keyword GCONINJE) item 10

**ECLIPSE 100 only**
- Sales gas control (keyword GCONSALE), which uses reinjection

**ECLIPSE 300 only**
- Advanced or Make-up gas (keywords GADVANCE or GINJGAS)

**ECLIPSE 300 only**
- Injection limited by availability (keyword WAVAILIM), because these facilities require the producers to be solved before the injectors.

**ECLIPSE 300**

If the composition of the injected gas stream is defined by the produced vapor from a group or well (keywords GINJGAS or WINJGAS), the composition will be taken from the previous converged Newton iteration.

The use of a production balancing control mode (that is, PRBL, PBGS or PBWS in item 2 of this keyword) only allows balancing of the production from a group with the injection from the same group. This behavior is in contrast with that of GCONINJE in which items 11 and 12 allow the injection to a group from the production from another group.

When a group which was operating under a rate target for one phase, switches to rate control by another phase, the guide rates for any children of this group are not allowed to be zero as explained in the note in "Group production rate limits" in the ECLIPSE Technical Description. Instead, a small positive guide rate is imposed on such wells to force them to stay open and operate at their potential. Unfortunately, sometimes this means that the original group gas rate target is exceeded. Because of this, it is strongly recommended that RATE is not used to control breakthrough in this way. Two possible alternatives are

- use an alternative procedure such as WELL or CON under GCONPROD in items 11 onwards, or:
- use a GUIDERAT formula than penalizes wells that see breakthrough.
Examples

Figure 3.7. A five level hierarchy

Example 1

The field has a target oil rate of 10,000 stb/day of oil, to be shared between the wells in proportion to their guide rates or potentials, subject to their individual rate and pressure limits.

```
GCONPROD
  FIELD ORAT 10000 /
```

Example 2

As Example 1, but PLAT-A, PLAT-B and SAT-B have water and gas production limits. If these limits are exceeded, the worst-offending wells are to be worked over.

```
GCONPROD
  FIELD ORAT 10000 /
  'PLAT*' NONE 1* 6000 20000 1* CON /
  SAT-B  NONE 1* 4000 10000 1* CON /
```

Example 3

As Example 2, but PLAT-A and PLAT-B are required to produce equal amounts of liquid, if possible. They are given equal guide rates for the liquid phase, set to an arbitrary value of 1000, say. Since they have been given guide rates, they are placed under rate control from the higher (field) level.

```
GCONPROD
  FIELD ORAT 10000 /
  'PLAT*' FLD 1* 6000 20000 1* CON 1* 1000 LIQ /
  SAT-B  NONE 1* 4000 10000 1* CON /
```

Example 4

As Example 3, but GR-A1 is required to produce as much as it can. The group is declared unable to respond to a higher level rate target. Its wells then produce as much as they can according to their own rate and pressure limits.

```
GCONPROD
  FIELD ORAT 10000 /
  'PLAT*' FLD 1* 6000 20000 1* CON 1* 1000 LIQ /
  SAT-B  NONE 1* 4000 10000 1* CON /
```

Keywords
Example 5

As Example 4, but in addition GR-B1 is required to produce 2000 stb/day of oil, irrespective of any higher level rate target. It is declared unable to respond to a higher level rate target, and placed under oil rate control with a target rate of 2000.

Example 6

The field is set to produce at a reservoir volume rate equal to the total reservoir volume injection rate of the field. The reservoir volume production rate target is to be shared between PLAT-A and PLAT-B in proportion to their individual reservoir volume injection rates.
GCONSALE  Sales gas production controls for groups/field

This keyword provides a means of controlling the export of associated gas from an oil field, which may at the same time have controls on the oil production rate. It is not intended for use when simulating gas fields, as it relies on gas injection to control the sales gas rate.

The sales gas production rate of a group is defined as

\[
\text{(sales gas production rate)} = (\text{total gas production rate}) - (\text{gas injection rate}) + (\text{gas import rate to the group and any subordinate groups}) - (\text{gas consumption rate of the group and any subordinate groups})
\]

The gas consumption and import rates are set with the keyword GCONSUMP.

The sales gas production rate is controlled by reinjecting the surplus gas not required for sale. Sales gas control can be applied independently of any other production controls on the group or field, provided that there is enough injection capacity to inject the surplus gas. There must be one or more gas injectors subordinate to each group with a sales gas target. The group or field is automatically placed under gas reinjection control, and its target reinjection fraction is determined dynamically at each timestep to inject the surplus gas. Other limits on gas injection for the group or field (for example maximum surface rate) can be simultaneously applied using the keyword GCONINJE; place the group under REIN control, with any value for the reinjection fraction, and supply the values for any other gas injection limits. The distribution of the injected gas between any subordinate groups can also be controlled by giving these groups upper rate limits and guide rates for gas injection, with the keyword GCONINJE.

If the group or field is producing more surplus gas than the injectors can handle, the drilling queue will be scanned for a new gas injector to open. Failing this, a choice of actions can be performed at the end of the timestep to reduce the gas production in the next timestep. The choice is governed by item 5 in this keyword. If on the other hand the group or field is not producing enough gas to fulfil the sales requirements, corrective action is taken at the end of the timestep, as described under item 4.

The keyword is followed by any number of records, each containing the following items of data and terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root
   
   or FIELD (for field sales gas control data)
   
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Sales gas production rate target. User defined arguments can be set with keywords UDQ and UDADIMS.

   A negative value switches off sales gas control (and reinjection) for the group. If you wish to continue reinjection, or impose other forms of group gas injection control, you must initiate this by subsequent use of the keyword GCONINJE.

   UNITS: \( \text{sm}^3/\text{day (METRIC)}, \text{Mscf/day (FIELD), scc/hr (LAB), sm}^3/\text{day (PVT-M)} \)

3. Maximum permitted sales gas production rate. User defined arguments can be set with keywords UDQ and UDADIMS.

   This value should be greater than the target rate in item 2.

   If the maximum rate is exceeded, the action requested in item 5 is taken at the end of the timestep.

   UNITS: \( \text{sm}^3/\text{day (METRIC)}, \text{Mscf/day (FIELD), scc/hr (LAB), sm}^3/\text{day (PVT-M)} \)
4. Minimum permitted sales gas production rate. User defined arguments can be set with keywords UDQ and UDADIMS.

UNITS: \( \text{m}^3/\text{day} \) (METRIC), Mscf/day (FIELD), scc/hr (LAB), \( \text{m}^3/\text{day} \) (PVT-M)

This value should be less than the target rate in item 2.

If the rate falls below the minimum, the corrective action described below is taken at the end of the timestep:

If The group is limited by a maximum gas production rate, increase its gas rate limit by the amount necessary to reach the sales gas target

Else If There is a special gas producer (see keyword WGASPROD) open subordinate to the group and able to increase its production rate, increase the target gas rate of the special gas producer by its incremental rate set in keyword WGASPROD

Else If There are more special gas producers in the drilling queue, open the next special gas producer in the drilling queue that is subordinate to the group needing more sales gas, and not subordinate to a group under gas rate control or a prioritization group, and set the well’s initial gas rate target equal to its incremental rate

Else If There are no suitable special gas producers in the drilling queue, open the next ordinary producer in the drilling queue that is subordinate to the group needing more sales gas, and not subordinate to a group under gas rate control or a prioritization group.

DEFAULT: Minus infinity (-1.0E20)

Note: Prior to 2007.1, the default for this item was 0 (Zero). If a non-positive lower limit value was specified, then this was interpreted as “no lower limit”. This was changed at 2007.1 in order to allow negative lower limit values to be specified. The pre-2007.1 behavior can be recovered by setting item 103 in OPTIONS to a positive value.

5. Procedure on exceeding a maximum rate limit

NONE Do nothing

CON Shut worst-offending connection in worst-offending well

+CON Shut worst-offending connection and all below it in worst-offending well

WELL Shut or stop worst-offending well

(item 9 in keyword WELSPECS)

PLUG Plug back worst-offending well (see keyword WPLUG)

RATE Reduce the group’s gas production rate target/limit to a value that meets the sales gas target after allowing for any consumption and the current rate of reinjection. Place the group on gas production rate control.

MAXR As RATE, but maximize the future production rate by setting the reinjection fraction limit to 1 (or 100 in cases with a net sales, import and consumption target of zero) so that if the
injection capacity subsequently increases the gas production target increases correspondingly. (See "Sales gas production control" in the ECLIPSE Technical Description.)

Pre-2007.1, the reinjection limit was set to 1 instead of 100 in cases where the net sales, import and consumption was zero. This had caused the MAXR algorithm to stagnate and not take advantage of any increases in injection capacity. The old behavior can be recovered by setting item 103 in OPTIONS to a positive value.

The new “zero net sales, import and consumption” target gas reinjection fraction limit value of 100 can be modified by setting OPTIONS item 103 to a negative value.

END     Stop the simulation at the end of the report step

DEFAULT: NONE

The ‘worst-offending’ well or connection here is the one producing the highest ratio of gas to the well’s preferred phase (item 6 of keyword WELSPCS). See also keywords COMPLUMP and WORKLIM.

The MAX option overwrites the group’s gas production rate limit with a value equal to the sales gas target plus the injection rate plus any gas consumption minus any import rate, while the group is operating on gas production rate control. Any user-defined limit on this quantity is erased.

The RATE and MAXR options are not allowed in groups using the prioritization option for production rate control (see keywords GCONPRI and PRIORITY).

Examples

Example 1

The field gas sales target is 50,000 Mscf/day. The surplus gas is to be reinjected by the gas injectors in proportion to their guide rates or injection potentials. If the surplus is too much for the injectors to handle, the worst-offending well will be shut in.

GCONSALE
FIELD  50000 55000 45000  WELL /
/

Example 2

As example 1, but the field is subject to a gas injection limit of 30,000 Mscf/day. The rate limit is imposed with the keyword GCONINJE. The field is placed under reinjection control (since this is the control mode for sales gas control), but the reinjection fraction set here will be ignored. The field will switch to gas production rate control if it is unable to reinject all the surplus gas.

GCONSALE
FIELD  50000 55000 45000  RATE /
/
GCONINJE
FIELD GAS REIN 30000 1* 1.0 /
/

Example 3

As example 1, but the keyword GCONINJE is used to control the distribution of reinjected gas. The grouping hierarchy is the same as that given in the examples for the GCONINJE keyword. The injected gas
is to be distributed between the bottom-level groups in proportion to their voidage production, subject to PLAT-A and PLAT-B not injecting more than their own available gas.

```
GCONSALE
   FIELD 50000 55000 45000 WELL /
/
GCONINJE
   'PLAT*' GAS NONE 2* 1.0 /
   'GR-*' GAS FLD 6* VOID /
/
```
This keyword is used to specify the rate of gas consumption and the gas import rate for individual groups. The gas consumption rate may represent, for example, gas used for power generation at a platform. The gas import rate can be used to take account of gas imported from outside the group.

Gas consumption and import rates are taken into account for group reinjection control (see keyword \texttt{GCONINJE}) and sales gas control (see keyword \texttt{GCONSALE}).

Under gas reinjection control, the gas injection rate is \( (\text{gas injection rate of group}) = (\text{the group’s reinjection fraction}) \times [(\text{the group’s gas production rate}) + (\text{the gas import rate to the group and any subordinate groups}) - (\text{the gas consumption rate of the group and any subordinate groups})] \)

The sales gas production rate is calculated as \( (\text{sales gas production rate}) = (\text{gas production rate}) - (\text{gas injection rate}) + (\text{gas import rate}) - (\text{gas consumption rate}) \)

Gas consumption and import rates are also taken into account by the network model (gas consumed is subtracted from the pipeline flows, gas imported is added - but see also item 4 below), and the gas field operations model (group/field target gas rates apply to sales gas rather than gross production from the formation).

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root
   
   or \texttt{FIELD} (if gas is consumed or imported at the field level)
   
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Gas consumption rate for the group
   
   If a negative value between 0.0 and -1.0 is specified, it is interpreted as minus the fraction of the group’s gas production. Thus for example a value of -0.1 sets the gas consumption rate for the group to one tenth of its total gas production rate. A negative value is not allowed when running the gas field operations model. A user defined argument can be set with keywords \texttt{UDQ} and \texttt{UDADIMS}.
   
   \begin{itemize}
   \item \textbf{UNITS}: \( \text{sm}^3/\text{day (METRIC), Msce/day (FIELD), scc/hr (LAB), sm}^3/\text{day (PVT-M)} \)
   \item \textbf{DEFAULT}: Zero
   \end{itemize}

3. Gas import rate for the group. The value must be greater than or equal to zero. A user defined argument can be set with keywords \texttt{UDQ} and \texttt{UDADIMS}.
   
   \begin{itemize}
   \item \textbf{UNITS}: \( \text{sm}^3/\text{day (METRIC), Msce/day (FIELD), scc/hr (LAB), sm}^3/\text{day (PVT-M)} \)
   \item \textbf{DEFAULT}: Zero
   \end{itemize}

4. Name of the network node from which the group’s consumption should also be removed or import added
   
   This item is only relevant if you are using the extended network model. It is ignored otherwise. The item defines the point in the network where the gas consumption should be subtracted from (or import added to) the network flow. This is necessary because the extended network structure is distinct from the group hierarchy, and in general there is no correspondence between groups and network nodes (except at the source-groups that supply production into the network, for example well-groups).

\textbf{Keywords}
If this item is defaulted when the extended network model is active, the result is as follows. If the group in item 1 is a source-group, then the default node is the node with the same name as the source-group. If the group in item 1 is not a source-group, then the default is that the consumption or import will not be taken into account in the network flows. If the group is a source-group but it is desired that the consumption or import rates are not accounted for anywhere in the network, then the character string 'NO-NODES' must be entered here.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Groups not specified in this keyword will have their gas consumption and import rates defaulted to zero.

**Example**

```
GCONSUMP
PLAT-A 20 50 /
PLAT-B 15 /
/```

**Keywords**
**GCONTOL**  
**Tolerance for group control targets**

This keyword provides a means of controlling the accuracy to which group production and injection calculations are performed. There are three issues concerning the accuracy:

- The number of Newton iterations of each timestep for which the well rate targets are calculated to meet the group targets (see notes 1 and 2).
- When the injection gas composition in ECLIPSE 300 depends on the injection rate, the composition must be calculated using an accurate estimate of the injection rate (see note 3).
- The number of injection iterations in ECLIPSE 300. For example, when the `WAVAILIM` keyword is entered in ECLIPSE 300 and a limited source of gas is shared between two or more injection groups, the injection calculation may have to be iterated a number of times to match the groups’ share of the available gas with their injection capability (see note 4).

**Note:** In ECLIPSE 300, the default values for items 3 and 4 are NOT the same as the defaults when GCONTOL is not included in this dataset. Use of this keyword with defaults for items 3 and 4 may change the simulation behavior for both gas and water injection. See note 5 for more details.

The keyword is followed by a line containing some or all of the following items, terminated with a slash (/).

1. **ECLIPSE 100 only**  
   Tolerance fraction to which group/field flow targets must be met (see note 2).
   
   If a target is not met to within the tolerance fraction times the target value, when there is sufficient potential to meet the target, the well flow rates are recalculated to meet the target regardless of the value of `NUPCOL`.
   
   The tolerance fraction does not apply to ECLIPSE 300, which ignores this value.
   
   DEFAULT: Infinity.

2. **Value of NUPCOL**, the number of Newton iterations in each timestep during which the well flow rates are recalculated to meet any collective control targets (see note 1).
   
   A value entered here overrides any value entered previously with keyword GCONTOL or NUPCOL.
   
   DEFAULT: Unchanged.

3. **ECLIPSE 300 only**  
   Tolerance fraction to which the most rapidly changing well gas injection rate estimate or well available gas fraction must be met (see notes 3 and 4).
   
   The injection calculation is repeated until the largest fractional change of all the well gas injection rate estimates lies within this tolerance, or the maximum number of iterations is reached (see item 4). If availability limits apply (see keyword WAVAILIM), this tolerance must also be met for the largest change of all the wells’ available gas fractions.
   
   DEFAULT: 1.0E-3.

4. **ECLIPSE 300 only**  
   Maximum number of iterations over the injection calculation (see notes 3 and 4).
   
   DEFAULT: 5.

**Notes**

1. When any form of group control option is being used (for example keywords GCONPROD, GCONINJE and GCONSALE), the flow targets of certain wells are directly influenced by the behavior of other
wells in the field. For example, the target rates of producers under group control must allow for the production from wells that are under other modes of control within the group. Also injection wells performing reinjection or voidage replacement have flow targets that depend on the behavior of the production wells.

For the group targets to be met exactly, the flow targets of the wells under group control must be recalculate at every Newton iteration. But when the well flow targets change at each iteration, the convergence rate of the iterations is reduced. This is because the Jacobian matrix calculation does not include the terms representing the mutual dependency of the well rates.

As a compromise, the well flow targets are updated in the first \texttt{NUPCOL} Newton iterations of each timestep, and are kept constant for any subsequent iterations. (This can be set at any time in the simulation with the keyword \texttt{NUPCOL} or item 2 of keyword \texttt{GCONTOL}.) Thus the group/field flow target is met exactly if the timestep converges within \texttt{NUPCOL} iterations. Any changes in the solution after the first \texttt{NUPCOL} iterations may cause the group/field flow rates to drift slightly away from their targets. In general, larger values of \texttt{NUPCOL} result in the group/field flow targets being met more accurately, but perhaps at the expense of requiring more Newton iterations to achieve convergence.

\textbf{ECLIPSE 100} 2. The discrepancy between a group/field flow target and its actual flow rate can be limited to lie within a tolerance fraction set in item 1 of this keyword. If at any Newton iteration a flow target is not met to within this tolerance, when there is sufficient capacity to meet the target, the well flow rates are recalculated to meet that target regardless of the value of \texttt{NUPCOL}. Since recalculating the well targets may retard the convergence of the Newton iterations, it is best not to make the tolerance too small. A lower limit of 0.01 is suggested.

The tolerance fraction does not apply when using the prioritization option (see keywords \texttt{GCONFRI} and \texttt{PRIORITY}). Also, the tolerance fraction currently does not apply to ECLIPSE 300.

\textbf{ECLIPSE 300} 3. The composition of the injected gas may vary with the injection rate when advance gas has been specified for the source group or well, or make-up gas has been specified for the injecting group or well (see keywords \texttt{GINJGAS} and \texttt{WINJGAS}), or when the injection gas is made up of a mixture defined by \texttt{WINJORD}. Thus an estimate of the injection rate is required in order to calculate the injection gas composition, before the injection calculation is performed. The actual injection rate may, in turn, depend on the composition of the gas, and whether additional constraints (for example, a well BHP limit) prevent a group from achieving its injection target. The injection calculation may have to be iterated a number of times until the actual injection rates agree with the estimates used to determine the composition, to a tolerance fraction specified in item 3. An upper limit to the number of these iterations can be set in item 4. An indication that the injection calculation has not yet converged would, for example, be a significant excess gas rate value reported for a group which is required to inject all its produced gas and some make-up gas in order to meet a reinjection target.

\textbf{ECLIPSE 300} 4. An iterative loop over the injection calculation is also required to obtain an accurate solution when two or more injection groups take their gas from the same source and are limited by availability (see keyword \texttt{WAVAILIM}). Each injection group is assigned a certain fraction of the available gas from the source. These fractions are initially calculated in proportion to the groups’ gas injection potentials. After each pass through the injection calculation loop, these fractions are recalculated using the actual gas injection rates. Thus, if a particular injection group injects significantly less than its allocated share of available gas, its unused share of available gas is reallocated to the other groups. It may take several loops around the injection calculation before the available gas fractions converge; items 3 and 4 may be used to control this iteration process.

\textbf{ECLIPSE 300} 5. items 3 and 4 control how the injection system is solved. The default values shown for items 3 and 4 are also the values adopted for the iterative loop solving the injection system if \texttt{GCONTOL} is absent and

- advance gas is used, or
• make-up gas is used, or
• a WINJORD mixture has been specified, or
• availability limits have been requested.

If none of these conditions are met, the injection calculation is only performed once per Newton iteration. If GCONTOL items 3 and 4 are defaulted, the limit on the number of Newton iterations is increased to 5.

Example
To set a two percent tolerance on group rate targets in ECLIPSE 100:

\[
\text{GCONTOL}
\begin{array}{l}
0.02 \\
3 \\
/ \\
\end{array}
\]

To limit the number of injection calculation iterations in ECLIPSE 300 to 2:

\[
\text{GCONTOL}
\begin{array}{l}
3* \\
2 \\
/ \\
\end{array}
\]
**Group cutback limits**

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

Default values can be specified by “null value” indicators (n*, where n is the number of consecutive items to be defaulted).

The set of records must end with a blank record, containing only a slash (/).

1. **Group name or group name root**
   
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. **Water cut upper limit for group rate cutback**
   
   **UNITS:** sm$^3$/sm$^3$ (**METRIC**), stb/stb (**FIELD**), scc/scc (**LAB**), sm$^3$/sm$^3$ (**PVT-M**)
   
   **DEFAULT:** Infinity (1.0E20)
   
   (A value of 0.0 or > 1.0 switches off this check)

3. **Gas-oil ratio upper limit for group rate cutback**
   
   **UNITS:** sm$^3$/sm$^3$ (**METRIC**), Mscf/stb (**FIELD**), scc/scc (**LAB**), sm$^3$/sm$^3$ (**PVT-M**)
   
   **DEFAULT:** Infinity (1.0E20)
   
   (A value of 0.0 or > 1E19 switches off this check)

4. **Gas-liquid ratio upper limit for group rate cutback**
   
   **UNITS:** sm$^3$/sm$^3$ (**METRIC**), Mscf/stb (**FIELD**), scc/scc (**LAB**), sm$^3$/sm$^3$ (**PVT-M**)
   
   **DEFAULT:** Infinity (1.0E20)
   
   (A value of 0.0 or > 1E19 switches off this check)

5. **Water-gas ratio upper limit for group rate cutback**
   
   **UNITS:** sm$^3$/sm$^3$ (**METRIC**), stb/Mscf (**FIELD**), scc/scc (**LAB**), sm$^3$/sm$^3$ (**PVT-M**)
   
   **DEFAULT:** Infinity (1.0E20)
   
   (A value of 0.0 or > 1E19 switches off this check)

6. **Rate cutback factor (fraction)**
   
   This is the ratio of the reduced flow rate to the current flow rate. It should be less than 1.0.
   
   **DEFAULT:** same as item 2 in the **GCUTBACT** keyword, if that has been entered previously, otherwise no default is allowed.

7. **Control phase, to which the reduced rate applies**
   
   - **OIL** Reduced oil rate
   - **WAT** Reduced water rate
   - **GAS** Reduced gas rate
LIQ  Reduced liquid rate

RESV Reduced reservoir fluid volume rate

COMB Reduced linearly combined rate (coefficients must be set with LINCOM)

You are advised to choose a phase that does not have a user defined rate limit, as it would be overwritten whenever a cutback or cutback reversal occurs.

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Whenever the group violates one of the cutback limits given in items 2 to 5, it is set to produce at a reduced target rate of the control phase specified in tem 7. The target rate of the control phase is set equal to the group’s current flow rate of that phase, multiplied by the cutback factor specified in item 6. The group’s production rate target is met by guide rate control (see keyword GCONFPROD), unless the group has been placed under prioritization with keyword GCONFPRI.

See also keyword WLIMITOL.

Example

```plaintext
GCUTBACK
G1  0.6 3*  0.9  LIQ /
G2  1*  3.0 2*  0.9  RESV /
/```

Keywords
**GCUTBACT Tracer cutback limits for groups**

*GCUTBACT* is followed by any number of records, described below, each record terminated by a slash (/).

The keyword is not compatible with the reservoir coupling option. A record may be terminated early at any item; the remaining items assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted. There are two types of record; one describes the connection and the second lists the tracers and their limits.

The set of records must end with a blank record, containing only a slash (/).

Records of the first kind can contain some or all of the following items:

1. **Group name or group name root.**
   
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. **Rate cutback factor (fraction).**
   
   This is the ratio of the reduced flow rate to the current flow rate. It should be less than 1.0. The same rate cutback factor applies to both this keyword and the *GCUTBACK* keyword. If *GCUTBACK* has been entered previously, this item overwrites item 6 in that keyword. If *GCUTBACK* is entered subsequently, its item 6 overwrites this item.

   **DEFAULT:** same as item 6 in the *GCUTBACK* keyword, if that has been entered previously; otherwise no default is allowed.

3. **Control phase, to which the reduced rate applies**
   
   **OIL** Reduced oil rate
   
   **WAT** Reduced water rate
   
   **GAS** Reduced gas rate
   
   **LIQ** Reduced liquid rate
   
   **RESV** Reduced reservoir fluid volume rate
   
   **COMB** Reduced linearly combined rate (coefficients must be set with *LINCOM*)

   You are advised to choose a phase that does not have a user defined rate limit, as it would be overwritten whenever a cutback or cutback reversal occurs. See also Note 5 in the *WCUTBACT* keyword.

End the record with a slash (/)

There can follow a maximum of three subsequent records for each group. These records have some or all of the following items.

1. **Tracer name (Use PLY for a polymer and BRI for brine).**
2. **Total tracer rate upper limit for well rate cutback**
   
   **DEFAULT:** No limit
3. **Total tracer rate lower limit for well rate cutback reversal**
   
   **DEFAULT:** No limit
4. Total tracer concentration upper limit for well rate cutback (not available for partitioned tracers).
   DEFAULT: No limit

5. Total tracer concentration lower limit for well rate cutback reversal (not available for partitioned tracers)
   DEFAULT: No limit

End the record with a slash (/).

End the data for this well with a blank line containing just a slash(/).

Note that this means that the keyword will end with two blank lines containing just slashes.

Notes

- No units are given for the tracer rates. The rates are those of the carrier flow.
- Since the tracers are solved after the main reservoir simulation, if it is subsequently discovered that a limit is violated the timestep is always repeated, with the action enforced from the start of the timestep. Even with doing this it may be that the ‘action’ lags the violation by a timestep.
- This keyword needs to end with two lines containing only slashes. See the example below.
- Whenever the group violates one of the cutback reversal limits, and if the group is currently operating under rate control of the phase selected in item 3 of record 1, the target rate of the phase is increased by dividing it by the factor specified in item 2. This essentially reverses the cutback process.

Example

```
GCUTBACT
  GRP1 0.8 WAT /
  TR1 1000.0 3*/
  TR2 2* 0.5 0.001 /
  TR3 1500.0 10.0 0.5 /
  / /
  GRP2 0.95 RESV /
  TR3 110.0 90.0 /
  / /
  GRP3 0.95 LIQ /
  TR4 2* 0.6 /
  PLY 2000 3*/
  / /
```
Initial coal gas concentration versus depth tables

The data comprises NTEQUL (keyword EQLDIMS in the RUNSPEC section) tables of initial gas concentration versus depth, one for each equilibration region. The keyword should only be used when the Coal Bed Methane option is active (keyword COAL in RUNSPEC section). Each table consists of 2 columns of data, and must be terminated by a slash (/).

1. Depth values. The values should increase monotonically down the column.

   **UNITS:** m (METRIC), ft (FIELD), cm (LAB)

2. The corresponding values of gas concentration in the coal.

   **UNITS:** sm$^3$/m$^3$ (METRIC), Mscf/ft$^3$ (FIELD), scc/cc (LAB)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NDRXVD (keyword EQLDIMS in the RUNSPEC section).

Both GASCONC and GCVD are entirely optional. If neither of these keywords is present, the coal gas concentration is taken as the equilibrium concentration for the corresponding fracture pressure.

The initial coal gas concentration should not be set above the initial equilibrium value; ECLIPSE issues a warning if it is. If the coal gas concentration is below the equilibrium value, the non-reversible diffusion model is recommended (set item 2 in keyword DIFFCOAL to 0.0).

See also the GASCONC keyword.

**Example**

With NTEQUL=1 and NDRXVD ≥ 2

```
GCVD
  7000  74.0
  10000 74.0 /
```

Keywords

910
**GDCQ**

**Initial DCQs for multiple contract groups**

This keyword must be used to set the initial DCQ values for each contract group, when running the Gas Field Operations Model in the multiple contract groups mode (initiated with keyword `GSWINGF`). The keyword must be entered after the contract groups have been defined with keyword `GSWINGF`, but before the first `GASYEAR` or `GASPERIO` keyword. The DCQ values can subsequently be reset at any time in the simulation by further use of the `GDCQ` keyword.

See "Applying separate gas supply contracts to individual groups" in the *ECLIPSE Technical Description* for further details.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. **Group name or group name root**
   - The group must previously have been defined as a contract group with keyword `GSWINGF`.
   - A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. **The initial DCQ (Daily Contracted Quantity) of the group**
   - **UNITS:** sm$^{3}$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^{3}$/day (PVT-M)
   - *ECLIPSE 100 only*
     - If item 1 of `DCQDEFN` is set to ‘ENERGY’:
       - **UNITS:** kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB), kJ/day (PVT-M)
       - **DEFAULT:** The group’s current DCQ

3. **Is the group’s DCQ variable or fixed?**
   - **VAR**  
     - ECLIPSE reduces the group’s DCQ as necessary to obey the swing requirement as defined in item 3 of keyword `GASYEAR` or item 4 of `GASPERIO`. (If there is more than one VAR contract group, only the options YEAR or PER and NO are allowed. The option YEAR or PER must be selected if you wish ECLIPSE to reduce the DCQs automatically.)
   - **FIX1**  
     - The group’s DCQ remains at the value set in item 2, even if the group cannot produce its target rate. During the first pass of each contract period, the group’s target rate is its DCQ × Swing Factor; or DCQ × Profile Factor if item 3 in keyword `GASYEAR` or item 4 in `GASPERIO` is PRO. During the second pass of each contract period, the group’s target rate is its DCQ × Profile Factor.
   - **FIX2**  
     - Same as for FIX1, but in addition a maximum gas production rate of DCQ × Swing Factor is imposed on the group while the delivery capacities are being calculated.

   - **DEFAULT:** VAR

End each data record with a slash (/).

End the set of records with a blank record, containing just a slash (/).

**Example**

```
GDCQ
PLAT-A  60000  VAR  /
```
PLAT-B 40000 FIX2 /
GDCQECON  Minimum economic DCQ limit for contract groups

The keyword sets a minimum economic DCQ for contract groups, when running the Gas Field Operations Model (see "Gas field operations model" in the ECLIPSE Technical Description).

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Contract group name or name root,  
   or FIELD (if the FIELD is the contract group)

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Minimum economic DCQ value

   If the DCQ falls below the specified minimum value, all the producers subordinate to the contract group will be shut or stopped (see item 9 in keyword WELSPECS). If the contract group is the FIELD, the run is terminated.

   A zero value switches off this limit.

   UNITS: sm$^3$/day (METRIC), Mcf/day (FIELD), ccf/hr (LAB), sm$^3$/day (PVT-M)

   ECLIPSE 100 only
   If item 1 of DCQDEFN is set to ‘ENERGY’:

   UNITS: kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB), kJ/day (PVT-M)

   DEFAULT: 0.0

Example

```
GDCQECON
FIELD  20000 /
/
```

Keywords

913
**Imports a grid file**

This keyword allows the grid geometry to be defined by an external Grid file. Typically, this could be a grid file generated by ECLIPSE, and it must have the required format of an ECLIPSE Grid file.

The external Grid file is composed of a set of topologically cuboidal cells, with the (X,Y,Z) position of each of the 8 corners specified independently. This allows a more flexible definition of the grid geometry than is available with the ZCORN and COORD keywords. The keyword also supports reading an EGRID format file.

See also the IMPORT keyword.

The keyword should be followed by the name of grid file to be imported and a file formatting flag:

1. Name of Grid file to be imported.
   
   This argument should include the file extension and is case sensitive, that is the string entered should exactly match the name of the file to be read. The complete file name, including extension, should not exceed 72 characters in length.

2. Formatted flag (optional).
   
   There are 2 choices:

   - FORMATTED  Formatted file
   - UNFORMATTED  Unformatted file

   The word may be shortened to F or U.

   DEFAULT: UNFORMATTED

   The data should be terminated by a slash (/).

**CAUTION:**

- This keyword should be used with care and is not recommended for general use. It is not possible to use it in radial geometry option, or with auto refinement.

- In ECLIPSE 100, the geometric data in the imported Grid file is skipped for any local grids

- In ECLIPSE 100, this keyword is also not recommended for use with the Vertical Equilibrium option.

---

**ECLIPSE 100**

*Note:* If **item 134** in **OPTIONS** is set to 1, a wider range of global cells is used as reference when calculating the geometry of the LGR cells. This can provide better output to the .EGRID file for visualization purposes.

**ECLIPSE 300**

Prior to the 2012.1 version, ECLIPSE 300 accepted the filename without its extension and then proceeded to search for either a matching .EGRID or .GRID file in this order. The file name was additionally converted to upper case. Whilst this is not the recommended usage of the keyword, the behavior is retained for backwards compatibility and is triggered by providing a string without a period ‘.’ as the first argument to the keyword. This is illustrated by Example 2.

---

**Keywords**
Examples

For the examples which follow, the simulator reads the geometric grid data from the file specified, after which it switches back to the next keyword in the current file.

Example 1

For ECLIPSE 100 and ECLIPSE 300 where the file to be read is ModelGrid.eGrid:

```
GDFILE
  ModelGrid.eGrid /
```

Example 2

For ECLIPSE 300 exercising the legacy behavior where the file to be read is either MODELGRID.EGRID or MODELGRID.GRID

```
GDFILE
  ModelGrid /
```
Specify the dimensions for the instantaneous gradient option

This keyword specifies the dimensions for the instantaneous gradient option, see the "Instantaneous gradients" section in the ECLIPSE Technical Description. The keyword is followed by a single item of data, terminated by a slash (/). The instantaneous gradient option requires an unencoded gradient license since the results are accessed as summary quantities.

1. The maximum number of well gradient parameters defined by the GWRTWCV keyword in the SCHEDULE section.
   DEFAULT: 0

Example

```
GDIMS 3 /
```
GDORIENT  Grid orientation

This is an optional keyword which data generation applications (and manually generated decks) can use to indicate to ECLIPSE and to post-processing software the orientations of the grid and the data prescribed on it. ECLIPSE itself will not use this information in its calculations, but it will attempt to verify that the Z-direction and handedness, values 4 and 5, are consistent with the grid. It will not modify or override the grid geometry specified with for example COORD or ZCORN regardless of the outcome of the verification.

Following the keyword line, there is a single record containing five items of data, ending with a slash (/):

1. Indicates property ordering, taking the value INC if properties such as PORO or PERMX are input in order of increasing I, otherwise it takes the value DEC.

   DEFAULT: INC

2. Indicates property ordering, taking the value INC if properties such as PORO or PERMX are input in order of increasing J, otherwise it takes the value DEC.

   DEFAULT: INC

3. Indicates property ordering, taking the value INC if properties such as PORO or PERMX are input in order of increasing K, otherwise it takes the value DEC.

   DEFAULT: INC

4. The Z-direction

   DEFAULT: DOWN

5. The handedness of the grid.

   DEFAULT: RIGHT

If the keyword is not supplied the simulator will assume default values for items 1-3 and attempt to determine items 4 and 5 directly from the grid.

Note that some pre- and post-processors do not support left-handed grids.

Example

```
GDORIENT
   INC INC INC DOWN RIGHT /
```
**GDRILPOT**

**Group minimum potential rates for drilling**

This keyword specifies a minimum potential flow rate for a group (see "Well potentials" in the *ECLIPSE Technical Description*). If the group’s production or injection potential falls below this limit the drilling queue (set up with either keyword **QDRILL** or **WDRILPRI**) will be scanned for a new well, of the appropriate type, to open to restore the group’s potential.

This feature allows the drilling process to be started in good time before the well is required to maintain a group production or injection target set with keyword **GCONPROD** or **GCONINJE**, by setting the group’s minimum potential rate sufficiently greater than its actual flow target, if the well has a non-zero drilling time set in item 2 of keyword **WDRILTIM**.

*Note:* If wells are treated as closed while being drilled (see item 3 of keyword **WDRILTIM**), *ECLIPSE* continues to drill new wells because the group potential will not increase immediately. This can be avoided by using the **QDRILL** option in the **WDRILTIM** keyword.

The **GDRILPOT** keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root
   - or **FIELD** (for field minimum potential rates)
     A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Quantity to which the rate in item 3 applies
   - **OPR** Oil production potential
   - **WPRD** Water production potential
   - **GPRD** Gas production potential
   - **OINJ** Oil injection potential
   - **WINJ** Water injection potential
   - **GINJ** Gas injection potential
   
   *ECLIPSE 100 only*
   
   **Note:** Only one production quantity can be active at any one time for each group, but injection limits may be defined simultaneously by entering multiple records for the same group.

3. Minimum potential rate of the quantity in item 2
   The value must be greater than or equal to zero.
   
   **UNITS:** sm$^3$/day **(METRIC)**, stb/day for oil or water **(FIELD)**, Mscl/day for gas **(FIELD)**, scc/hr **(LAB)**, sm$^3$/day **(PVT-M)**
   
   **DEFAULT:** Zero

---

**Keywords**

918
A zero or negative value implies that there is no minimum potential rate, for the quantity specified in item 2, applying to this group.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Example

```
GDRILPOT
-- group
  -- name  phase  rate
  GR-A    OPRD    7520  /
  GS-A    GPRD    17760 /
  GR-A    WINJ    3000  /
  GR-A    GINJ    35000 /
/
```

Keywords

919
Economic limit data for groups and the field

The keyword is followed by any number of records, each containing the following items of data, and each
terminated with a slash (/).

The records may be terminated at any item after item 1. The remaining items assume their default values.
Default values can be specified before the slash by a null repeat count of the form n*, where n is the
number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root
   or FIELD (for field economic limit data)
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several
groups in one record.

2. Minimum oil production rate. User defined arguments can be set with keywords UDQ and UDADIMS.
   UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)
   DEFAULT: 0.0
   If the group (or field) oil production rate falls below the specified minimum, all the producers in the
group (or field) are shut or stopped (see item 9 in keyword WELSPECS). A zero or negative value
switches off this constraint.

3. Minimum gas production rate. User defined arguments can be set with keywords UDQ and UDADIMS.
   UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)
   DEFAULT: 0.0
   If the group (or field) gas production rate falls below the specified minimum, all the producers in the
group (or field) are shut or stopped (see item 9 in keyword WELSPECS). A zero or negative value
switches off this constraint.

4. Maximum group or field water cut (water-liquid ratio). User defined arguments can be set with
   keywords UDQ and UDADIMS.
   UNITS: sm$^3$/sm$^3$ (METRIC), stb/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)
   DEFAULT: No limit
   (A value of 0.0 switches off this constraint).

5. Maximum group or field gas-oil ratio. User defined arguments can be set with keywords UDQ and
   UDADIMS.
   UNITS: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)
   DEFAULT: No limit
   (A value of 0.0 switches off this constraint)

6. Maximum group or field water-gas ratio. User defined arguments can be set with keywords UDQ and
   UDADIMS.
   UNITS: sm$^3$/sm$^3$ (METRIC), stb/Mscf (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)
7. Workover procedure on exceeding water cut, GOR or WGR limit

NONE  Do nothing

CON  Shut worst-offending connection in worst-offending well

+CON  Shut worst-offending connection and all below it in worst-offending well (see keyword COMPORD)

WELL  Shut or stop worst-offending well (item 9 in keyword WELSPECS)

ECLIPSE 100 only  PLUG  Plug back worst-offending well (see keyword WPLUG)

ECLIPSE 300 only  ALL  Shut all producers subordinate to the group

DEFAULT: NONE

8. End run flag

YES  The run stops at the next report time if all the producers in the group (or the field) are shut or stopped for any reason, provided the group has had at least one producer open previously

NO  The run continues regardless

DEFAULT: NO

ECLIPSE 100

Note: If the GCALECON keyword of the gas calorific value control option is entered subsequently to the GECON keyword, item 4 of the GCALECON keyword overwrites the ‘end run flag’ defined here.

ECLIPSE 100 only  9. Maximum allowed number of open wells subordinate to this group

DEFAULT: 0 (Which implies no limit)

This option prevents wells from being opened automatically (AUTO in keywords WCONPROD, WCONINJE, WELOPEN) or from the drilling queue (keyword QDRILL) or swapped from one group to this one (keyword WREGROUP) if the number of subordinate wells would violate this limit. The limit does not prevent wells being opened manually (‘OPEN’ in keywords WCONPROD and WCONINJE for example), but a warning message is issued if they violate this limit.

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Note: The actions taken as a result of the GECON keyword are applied at the end of the timestep. The run normally continues with the next timestep. However, the WLIMTOL keyword may be used to instruct the simulator to redo the timestep if an economic limit has been violated by more than a specified tolerance.

See also the ECLIPSE 100 keywords COMPLUMP and WORKLIM.
Example

GECON
GRUP1 2000 1* 0.8 10.0 1* CON Y /
GRUP2 1000 /
FIELD 4000 5* Y /
/
GECONT Tracer economic limits for groups

GECONT is used to set tracer economic limits for groups, including polymer and brine. The keyword is not compatible with the reservoir coupling option.

GECONT is followed by any number of records, described below, each record terminated by a slash (/). A record may be terminated early at any item; the remaining items assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted. There are two types of record; one describes the connection and the second lists the tracers and their limits.

The set of records must end with a blank record, containing only a slash (/).

Records of the first kind can contain some or all of the following items:

1. **Group name or group name root**
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. **Workover procedure when a limit is violated**
   - NONE No action to be taken
   - CON Shut worst-offending connection
   - +CON Shut worst-offending connection and all those below it (see keyword COMPORD)
   - WELL Shut or stop the worst-offending well (item 9 in keyword WELSPecs)
   - PLUG Plug back the worst offending well (see keyword WPLUG)
   
   **DEFAULT:** NONE

3. **End run flag**
   - YES The run stops at the next report time if the well is shut or stopped for any reason after being opened
   - NO The run continues regardless
   
   **DEFAULT:** NO

4. **Maximum allowed number of open wells subordinate to this group**
   **DEFAULT:** 0 (which implies no limit)
   
   This option prevents wells from being opened automatically (AUTO in keywords WCONPROD, WCONINJE, WELOPEN) or from the drilling queue (keyword QDRILL) or swapped from one group to this one (keyword WREGROUP) if the number of subordinate wells would violate this limit. The limit does not prevent wells being opened manually ('OPEN' in keywords WCONPROD and WCONINJE for example), but a warning message is issued if they violate this limit.

End the record with a slash (/).

A maximum of three subsequent records can follow, for each group. These records can contain some or all of the following items.
1. Tracer name (Use PLY for a polymer and BRI for brine)

2. Maximum total tracer rate
   DEFAULT: No limit

3. Maximum total tracer concentration (not available for partitioned tracers)
   DEFAULT: No limit
   The following are only available for two-phase tracers.

4. Maximum free tracer rate
   DEFAULT: No limit

5. Maximum free tracer concentration
   DEFAULT: No limit

6. Maximum solution tracer rate
   DEFAULT: No limit

7. Maximum solution tracer concentration
   DEFAULT: No limit.

End the record with a slash (/).

End the data for this well with a blank line containing just a slash(/).

Note that this means that the keyword will end with two blank lines containing just slashes.

Notes

- No units are given for the tracer rates. The rates are those of the carrier flow.
- Due to the tracers being solved after the main reservoir simulation if it is subsequently discovered that a limit is violated the timestep is always repeated with the action being enforced from the start of the timestep. Even with doing this it may be that the ‘action’ lags the violation by a timestep.
- This keyword needs to end with two lines containing only slashes. See the examples.

See also keyword WLIMTOL and the ECLIPSE 100 keyword WORKLIM.

Examples

Example 1

```plaintext
GECONT
  GRP1  CON   /
  TR1 1000.0 0.5 /
  TR2 2* 1* 0.8 2* /
  TR3 1500.0 0.1 /
/
  GRP2  CON+ /
  TR3 100.0 0.05 /
/
  EAST1 WELL /
  TR4 200 5* /
  PLY 1* 0.7 /
/
```

Keywords

924
Example 2

GECONT
   NORTH CON /
   TR5 1000.0 0.5 /
   TR1 2* 1* 0.8 2* /
   TR3 1500.0 0.1 /
/
/

Example 3

GECONT
   SOUTH CON+ /
   BRI 1000.0 0.1 /
/
/
GEFAC

Sets group efficiency factors (for downtime)

The keyword allocates efficiency factors to groups, rather than individual wells. Group efficiency factors may be applied when all the wells subordinate to a group have their downtimes synchronized, so that the group either flows with its full rate or not at all. If the wells are taken down individually, rather than all together, efficiency factors should be allocated to the wells (with keyword WEFAC) instead of the group.

The GEFAC keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

   Note: The FIELD cannot have an efficiency factor.

2. Efficiency factor for the group

   This is the fractional time for which the group is operational. For example, if the group is down for 10 percent of the time, its efficiency factor is 0.9.

   The efficiency factor must be greater than zero.

   DEFAULT: 1.0

3. Should the group’s efficiency factor be automatically transferred to its corresponding node in the Extended Network option? (See "Well and group efficiency factors" in the ECLIPSE Technical Description).

   YES The node’s flow contribution to the flow rate of its parent node in the network will be multiplied by the group’s efficiency factor. Thus the branch pressure losses downstream of the parent node reflects the time-averaged contribution of this group. (Note that the pressure losses in the outlet branch of this group’s node still reflect the full, or reported, group flow rate.)

   NO The node’s contribution to the flow rate of its parent node in the network is not multiplied by the group’s efficiency factor. Thus all downstream branch pressure losses reflect the full flow rate of this group.

   This flag applies only to the Extended Network Model. If the Standard Network Model is being run, the groups’ reported flows are always used to calculate the pressure losses in the branch to their parent group.

   The flag also applies only if there is a node in the extended network that has the same name as the group. Otherwise, the NEFAC keyword can be used to apply efficiency factors directly to the network nodes.

   DEFAULT: YES

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

The efficiency factors for groups not specified with this keyword are defaulted to 1.0.

Keywords
Treatment of group downtime

If wells are brought down individually on a regular basis for short periods of maintenance, each should be given its own efficiency factors with keyword \texttt{WEFAC}. The treatment of well downtime is described in the \texttt{WEFAC} keyword documentation.

If all the wells in a group are brought down together for regular maintenance, then the efficiency factor should be applied to the group. The treatment of group downtime is entirely analogous to the treatment of well downtime. The group is solved with its full flow rate, which is the sum of its wells’ full flow rates (unless any wells or subordinate groups also have efficiency factors allocated). The group’s flow rates reported in the Print and Summary files also reflect its full flow rate. Any flow targets and limits for the group will apply to its full flow rate. But the group rates are multiplied by the group efficiency factor when summing the group’s contribution to its parent group in the hierarchy. Thus the flows of any superior groups will reflect this group’s time-averaged flow rate.

Below the group, the flows of the wells and any subordinate groups reflect the full flowing rates (subject to any efficiency factors applied at a lower level). But the well and group flows are multiplied by the efficiency factor when incrementing their cumulative flows over the timestep, and the group efficiency factor is also applied to the subordinate well connections when determining the source/sink terms in the reservoir grid blocks.

If the Standard Network option is in use, the groups’ \texttt{reported} flows are always used to calculate the pressure losses in the branch to their parent group. Thus if a group (but none of its subordinate groups or wells) has an efficiency factor set < 1.0, the flow in the branch to its parent group, and any branches below it in the group tree, reflects the full flow rate, while the flow in its parent group’s network branch and any branches above it in the group tree reflects the group’s time-averaged rate.

The Extended Network option has more flexibility in handling downtime. If an efficiency factor is applied to a group that has the same name as a node in the network, by default the efficiency factor is also applied to the corresponding node. The node’s flow rates are multiplied by the efficiency factor when adding them to the flow in its parent node. The pressure drop in a node’s outlet branch is calculated from the node’s flow rate. Thus the node’s outlet branch and the branches upstream of the node reflects the full flow rates, while the branches downstream of its parent node reflects the time-average flows. (This is equivalent to the Standard Network Model’s treatment of group efficiency factors.) However, by setting item 3 of this keyword to \texttt{’NO’} it is possible to specify that the group’s full flow rates should be used for calculating all downstream pressure losses, by not assigning the group’s efficiency factor to the corresponding network node.

If the wells are receiving gas lift, their gas lift injection rates reflects the full flows. But the group’s contribution to the lift gas supply rate of its superior groups are multiplied by its efficiency factor to reflect the time-averaged rate.

\textbf{Note:} It is not normal for efficiency factors to be allocated on more than one level of the well/group hierarchy. If they are, then the efficiency factors are multiplicative. That is, if a group has an efficiency factor of 0.9 (implying that all its wells are down together for 10 percent of the time), and also one of its wells is given an efficiency factor of 0.9 in \texttt{WEFAC}, then that well’s contribution to the field’s production rate and its cumulative totals is reduced to 81 percent of its full rate. A well’s efficiency factor product (multiplied with the efficiency factors of all its superior groups) may be written to the Summary file using the \texttt{SUMMARY} section keyword \texttt{WEFFG}.

\textbf{Note:} If you are using the Reservoir Coupling option, there are some subtle issues related to the use of the group efficiency factor.
Due to possible differences in timestepping between the Master and Slave simulations, the actual mechanism for passing rates from the Slave to the Master is to pass the cumulative flow, then the Master calculates the difference in cumulative flow over its timestep to work out the flow rates. The efficiency factor from the Slave group is also passed to the Master so that its effect can be removed, allowing the master group to calculate the peak flow. The Master group cumulative values are subsequently calculated from the calculated peak flow, the Master group efficiency factor and the Master group timestep.

**If you want the Master group’s time averaged rate to be based on the same efficiency factor as the Slave, you need to re-specify the Slave group efficiency in the Master group.**

In some situations, you may need to specify different efficiency factors for the Master group and Slave group, for example if you have an efficiency factor in the group hierarchy *above* the Master group, then this can be specified for the Slave group so that it reports the correct peak and averaged flows.

One further issue to be aware of is that Master group cumulative quantities will revert to the Slave cumulative quantity if a Master timestep is chopped.

### Example

```
GEFAC
PLAT-A 0.9 /
PLAT-B 0.8 /
```

Geomechanics dimension data

The data consists of the following items, describing the dimensions of geomechanics data to be used in the run. This keyword is only valid when the GEOMECH keyword is chosen in the RUNSPEC section. The data must be terminated by a slash (/).

1. Number of geomechanics regions in the model (NGMYRG).
   DEFAULT: 1
2. Number of rows in a PERMSTAB table (NGMPTR).
   DEFAULT: 1
3. Number of columns in a PERMSTAB table (NGMPTC).
   DEFAULT: 1
4. Number of rows in a BIOTCTAB table (NGMBTR).
   DEFAULT: 1
5. Number of columns in a BIOTCTAB table (NGMBTC).
   DEFAULT: 1

Example

```
GEODIMS
1 3 4 3 4 /
```
Geomechanics calculation

A rock stress (force) conservation in each coordinate direction is calculated in addition to the fluid mass and energy conservation.

The keyword is followed by a single record that contains options on the frequency of running the coupled geomechanics calculation and the type of calculation.

1. This item is not used.
2. A character specifying how often the coupled geomechanics calculation is run
   - R run the geomechanics calculation at each report step
   - T run the geomechanics calculation at the end of each timestep
   DEFAULT: R
3. A character string requesting the stress calculation method
   - FE requests a finite element stress calculation
   DEFAULT: FE
4. A character string specifying the porosity-stress relationship model
   - RVBAL requests that rock volume be conserved
   - RMBAL requests that rock mass be approximately conserved
   - RCOMP requests that porosity is determined from a ROCKCOMP model
   DEFAULT: RVBAL

Note: The rock volume conservation model is implemented with the relation
\[ V^{n+1}(1-\varphi^{n+1}) = V^n (1-\varphi^n) \],
which can also be written \( \delta\varphi = -(1-\varphi)\delta\varepsilon_b \) where \( V \) is the bulk volume, \( \varphi \) is the porosity, \( n \) is the timestep number and \( \delta\varepsilon_b \) is the change in volumetric strain. The approximate rock mass conservation model is implemented following the relation
\[ \delta\varphi = -\left(1 - \varphi - \frac{C_r}{C_{bc}}\right)(\delta\varepsilon_b - C_r\delta P) \]
where \( C_r \) is the rock or matrix compressibility, \( C_{bc} \) is the bulk compressibility, \( \varepsilon_b \) is the bulk volumetric strain, and the symbol \( \delta A \) denotes \( A^{n+1} - A^n \) where \( A \) is any quantity that changes from timestep \( n \) to timestep \( n+1 \). Further details of this second relationship can be found in the discussion of the keyword ROCK and see also [Ref. 46] and [Ref. 134].

5. A character string specifying the initial equilibration method
   - INIT will preserve initial pore volumes and pressures after application of traction, stress or velocity boundary conditions.
   - NOINIT allows the reservoir to compress and change these quantities.
   DEFAULT: INIT
6. A character string specifying the sign of compressive stresses. All boundary tractions and stresses will follow this convention.

**COMPN**  Compressive stress is negative: \( \sigma^T = \sigma^e - \alpha P \)

**COMPP**  Compressive stress is positive: \( \sigma^T = \sigma^e + \alpha P \)

**DEFAULT:** COMPN

7. A character string specifying an initial thermal stress calculation

**INIT**

**NOINIT**

**DEFAULT:** NOINIT

The record can be terminated early with a slash (/), the remaining items taking default values.

Geomechanics in ECLIPSE 300 is discussed in the “Geomechanics” chapter of the *ECLIPSE Technical Description*.

**Note:** Some data must be entered in order for the coupled geomechanics calculation to proceed. The minimum data required is Young’s modulus as discussed in the keyword **YOUNGMOD** and the Poisson ratio as discussed in **POISSONR**.

**Example**

```plaintext
-- Run the coupled geomechanics finite element stress calculation,
-- update at each report step
-- rock volume conservation porosity-stress relation
-- initialize the solution to preserve specified pressures,
-- pore volumes and FIP

GEOMECH
```

**Keywords**

931
**GEOYCAP**

**Geomechanics cap model parameters**

This keyword is used to specify the extended Sandler-Rubin cap model to simulate the plastic deformation having a shear failure surface together with a cap shaped yield in compression.

The keyword is only valid if **GEOMECH** is specified in the RUNSPEC section. It allows the user to specify the cap model parameters region by region. Every record line corresponds to a geomechanical material region defined in **YLDNUM**, each of which is terminated by a slash (/). Each record consists of some or all of the following items:

1. Initial rock cohesive stress $C_0$
   - This must be a positive quantity.

2. Internal friction angle
   - **UNITS**: degrees (METRIC), degrees (FIELD), degrees (LAB), degrees (PVT–M)
   - **DEFAULT**: 30 degrees

3. UCS - Uniaxial Compression Strength
   - This must be a positive quantity

4. Material parameter $K$ that defines the ratio of the yield stress in triaxial extension to the yield stress in triaxial compression.
   - **DEFAULT**: 1.0

5. Ratio of principal ellipse radii $R$ for cap surface
   - **DEFAULT**: 1.0

6. Material parameter $W$ for hardening law
   - This must be a positive quantity.

7. Material parameter $D1$ for hardening law
   - This must be a positive quantity.

8. Material parameter $D2$ for hardening law
   - This must be a positive quantity.
   - **DEFAULT**: 0.0

9. Initial cap position $X_0$
   - This must be a non-positive quantity.
   - **DEFAULT**: 0.0

**Example**

```plaintext
-- FIELD units
GEOYCAP
-- Co PHI UCS K R W D1
1450 30 3600 1.0 3.2 0.13 9.79E-5 2*/
```

**Keywords**

932
**GEOYLDN**

**Geomechanics yield function parameters**

The keyword is only valid if **GEOMECH** is specified in the **RUNSPEC** section. It allows the user to specify the yield function parameters region by region. Every record line corresponds to a geomechanical material region defined in **YLDNUM**, each of which is terminated by a slash (/). The data is terminated by a null record (that is a record with no data before the terminating slash). Each record consists of some or all of the following items:

1. Initial rock cohesive strength, $C_0$
   
   This must be a positive quantity.

2. The angle of internal friction
   
   **UNITS:** degrees (METRIC), degrees (FIELD), degrees (LAB), degrees (PVT-M)
   
   **DEFAULT:** 30 degrees

3. The failure model
   
   This must be one of:
   
   - **MC** indicating the Mohr-Coulomb model
   - **DP** indicating the Drucker-Praeger model.
   
   **DEFAULT:** **MC** (Mohr-Coulomb)

4. UCS - uniaxial compressive strength
   
   This must be a positive quantity

5. Type of hardening law.
   
   This must be one of:
   
   - **'HYP** indicating a hyperbolic law
   - **'POW'** indicating a power hardening law
   - **'PER'** which selects perfect plasticity.
   
   **DEFAULT:** **'PER'**

6. ‘a’ parameter for hardening - see equations below

7. ‘b’ parameter for hardening - see equations below

The hardening laws are implemented as a change in the shear cohesive strength, $C_0$, with change in generalized plastic strain $\bar{\varepsilon}_p$.

Generalized plastic strain is defined as

$$\bar{\varepsilon}_p = \sum_{i,j} \frac{\bar{\varepsilon}_{ij}^p \cdot \sigma_{ij}}{F}$$

where
\( \varepsilon_{ij}^p \) is the plastic strain vector,

\( \sigma_{ij} \) is the effective stress tensor and

\( F \) is a norm of the stress tensor.

The power law takes the form \( C_o = a \varepsilon_{ij}^b + C o_1 \) while the hyperbolic law follows the relation

\[
C_o = \frac{\varepsilon_p}{a + b \varepsilon_p} + C o_1.
\]

**Note:** Only one of Initial Rock Cohesive Strength or Uniaxial Compressive Strength is necessary to define the yield function. If UCS is entered, it will be converted to Initial Rock Cohesive Strength. If both are entered, the value of Initial Rock Cohesive Strength will be used.

**Example**

In the PROPS section:

```
-- FIELD units
GEOYLDF
  1450 30 'MC' 3600 'HYP' 1 1 /
```
GETDATA  Read named array data from an ECLIPSE restart or init file

The GETDATA keyword allows ECLIPSE to read named arrays from previously generated initial or restart files. The keyword should be followed by four items of data:

1. Full name of the file from which to import.
   The file name should not exceed 72 characters in length.
2. Flag indicating whether the import file is formatted or unformatted (binary).
   A single word, either FORMATTED or UNFORMATTED.
   The word may be shortened to F or U.
   DEFAULT: UNFORMATTED
3. Name of the array (ZNAME) that you wish to construct by importing data from the import file.
   ZNAME must be a valid ECLIPSE array name for the current section of the dataset.
4. Name of the corresponding array (ZALT) in the import file.
   The array ZALT in the import file is imported into the simulation as the array ZNAME.
   DEFAULT: The same name as in item 3.

The data should be terminated by a slash (/).

Example

In this example, we want to input the SWCR array from the run old_run.init as SWATINIT in the current realization:

```
GETDATA old_run.init   UNFORMATTED   SWATINIT   SWCR /
```

Note

In order to obtain the correct information from the restart or init file, this keyword also requires the Grid file.

The “standard” Grid file is required (see GRIDFILE keyword). By default, GRIDFILE will produce an extensible Grid file (.EGRID) but not the “standard” Grid file. Before GETDATA is used, the base run, from which the restart files are obtained, must contain the GRIDFILE keyword with the first item set to 2. This will produce the correct Grid file needed to process the GETDATA keyword. This information needs to be read in or imported for the current run. Refer to keywords ACTNUM and GDFILE.
GETGLOB  

Read only the global restart data

This keyword can be used to instruct ECLIPSE to read in just the global grid data from a restart file of a previous run that had local grids present. Any data for local grids on the restart file are ignored. This may be useful, for example, if the local grids are no longer needed in the restart run.

The keyword has no associated data.

ECLIPSE 100. If the restart run has any local grids present, then these are treated as being added at the restart time. The run is initialized by inheriting the solutions from the global host cells at the restart time.

**Example**

```
ECLIPSE Reference Manual

---

Keywords

---

936
```
GETSOL

Obtain initial solution arrays from restart file

This keyword enables explicit initial state data, such as PRESSURE, SWAT, SGAS, XMF, YMF, RS, RV and PDEW, to be obtained from an ECLIPSE 100 type Restart file. The mnemonic required, the Restart file name and the report step index required, should be specified. The result of a GETSOL operation is exactly the same as specifying the same data with the corresponding keyword. The RPTRST keyword in ECLIPSE 100 may be used to request that bubble and dew points are written to the Restart file. (The fifth item in RPTRST should be set to 1). The Restart files to be read may be multiple or unified. If multiple Restart files are to be read the MULTIN keyword should be used.

Possibilities are:

- PRESSURE: Pressures
- SWAT: Water saturations
- SGAS: Gas saturations
- RS: Bubble point pressures
- RV: Vapor oil-gas ratios
- PDEW: Dew point pressures
- XMF: Liquid mole fractions
- YMF: Vapor mole fractions
- TEMP: Temperatures (for the Thermal option)

Mole fractions can only be obtained from a Restart file produced by ECLIPSE 300, using the RPTRST keyword. Usually, when using ECLIPSE 100 Restart files, the mole fractions are supplied using the XMF or XMFVP and YMF or YMFVP keywords.

Example

```
GETSOL
 PRESSURE BASE 1 /
 SWAT BASE 1 /
 SGAS BASE 1 /
```

Note

In order to obtain the correct information from the restart file, this keyword also requires the Grid file.

The “standard” Grid file is required (see GRIDFILE keyword). By default, GRIDFILE will produce an extensible Grid file (.EGRID) but not the “standard” Grid file. Before GETSOL is used, the base run, from which the restart files are obtained, must contain the GRIDFILE keyword with the first item set to 2. This will produce the correct Grid file needed to process the GETSOL keyword.

Keywords

937
**GI**

**Initial Gi values for enumeration**

The keyword may be used when the initial conditions are set by enumeration, to specify the initial Gi values in each grid block. If the keyword is not present, the initial Gi values will be set to zero by default.

The keyword cannot be used when the initial conditions are set by equilibration (keyword EQUIL). In equilibrated starts, the initial Gi values are all set to zero.

The keyword should be followed by one real number for every grid block, specifying the initial Gi value. The set of numbers must be terminated with a slash (/).

**UNITS:** sm$^3$/rm$^3$ (METRIC), Mscf/rb (FIELD), scc/rcc (LAB)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*7200). Note that spaces must not be inserted on either side of the asterisk (*).

See also the keywords PRESSURE, RS, RV, SWAT, PRVD and SGAS in the SOLUTION section.

See section entitled "Gi pseudo-compositional model" in the ECLIPSE Technical Description for more information.

**Example**

With NDIVIX=10, NDIVIY=1, NDIVIZ=1 specified in RUNSPEC keyword DIMENS:

```
GI
10*0.2 /
```

See section entitled "Gi pseudo-compositional model" in the ECLIPSE Technical Description for more information.
Variation of saturated properties with pressure and Gi

This keyword is an alternative method of specifying the variation of $R_v$, $R_s$, $B_g$ and $B_o$ with pressure and Gi. GIALL replaces the RSGI, RVGI, BOGI and BGGI keywords with the constraint that for all four properties the same pressure nodes are used. This can be a more convenient format when the data is supplied by the PVT $i$ package.

The data comprises NTPVT (item 2 in keyword TABDIMS) tables of multiplier functions, each terminated by a slash(/), describing the variation of saturated $R_v$, $R_s$, $B_g$ and $B_o$ with pressure and Gi value. Each table consists of up to NPPVT sets of data records (item 4 in keyword TABDIMS). Each set of records within a table consists of up to NPPVT sets of four values.

These four columns define two-dimensional ($P$, Gi) functions that multiply the saturated values of $R_v$, $R_s$, $B_g$ and $B_o$ at ($P$, Gi=0) to give the true saturated value at ($P$, Gi).

The first record of each elementary set of records contains the following items of data:

1. The oil pressure value.
   - The values should increase monotonically down the column.
   - UNITS: barsa (METRIC), psia (FIELD), atma (LAB)
2. The gas $R_v$ multiplying factor at a Gi value specified as the first item in the GINODE table.
3. The oil $R_s$ multiplying factor at a Gi value specified as the first item in the GINODE table.
4. The gas $B_g$ multiplying factor at a Gi value specified as the first item in the GINODE table.
5. The oil $B_o$ multiplying factor at a Gi value specified as the first item in the GINODE table.

For record 2 and all subsequent records up to a maximum of NPPV, there should be only 4 data item in the record. These items define the corresponding factors for the Gi values specified in the GINODE keyword.

See "Gi pseudo-compositional model" in the ECLIPSE Technical Description for further information.

Example

With NTPVT =1 and NPPVT =2

<table>
<thead>
<tr>
<th>--</th>
<th>RV</th>
<th>RS</th>
<th>BG</th>
<th>BO</th>
</tr>
</thead>
<tbody>
<tr>
<td>GIALL</td>
<td>100.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td></td>
<td>0.97</td>
<td>0.93</td>
<td>0.98</td>
<td>0.93</td>
</tr>
<tr>
<td></td>
<td>200.0</td>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td></td>
<td>0.952</td>
<td>0.941</td>
<td>0.927</td>
<td>0.898</td>
</tr>
<tr>
<td></td>
<td>/</td>
<td>/</td>
<td>/</td>
<td>/</td>
</tr>
</tbody>
</table>
Enable GI pseudo-compositional option

This indicates that the ECLIPSE 100 Gi pseudo-compositional model for gas condensate / volatile oil systems is required. The model is described in "Gi pseudo-compositional model" in the ECLIPSE Technical Description.

The keyword has no associated data.
GINJGAS  Specify the nature of the injected gas

This keyword specifies the nature of the gas to be injected by a group. This should be used if GCONINJE or GPMAINT is used to specify group gas injection control data in compositional models. However, it is not necessary to use GINJGAS for groups subordinate to a higher level group having gas injection controls for which the nature of its injected gas is defined; this ensured that the injection stream is that defined for the higher level group.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1.  The name of the group.
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2.  A character string specifying the nature of the injected gas.
   - GAS  The composition of the injected gas is set to that of the field separator gas. However, it is generally better to specify the injected gas composition more precisely. From 2007.1, GAS is internally converted to GV (for the field separator) which has better reinjection and gas accounting behavior. item 113 in OPTIONS3 can be used to recover the pre-2007.1 behavior.
   - STREA  The molar composition of the injected fluid has been defined using the WELLSTRE keyword. The name of the stream must be entered in item 3.
   - MIX  The molar composition of the injected fluid has been defined as a mixture using either keyword WINJMIX or WINJORD. The name of the mixture must be entered in item 3.
   - GV  The injected fluid is to be taken from the vapor production of a nominated group. The name of the group must be entered in item 3.
   - WV  The injected fluid is to be taken from the vapor production of a nominated well. The name of the well must be entered in item 3.
   - GRUP  The injected fluid is that defined for a superior group.
   
   DEFAULT: GRUP
   Only the first two characters are significant.

3.  A character string is required if the composition of the injection fluid was specified using STREAM, MIX, GV or WV in item 2.
   In these cases this item is a character string specifying the name of the wellstream, mixture, group or well that defines or supplies the injection stream.

   The available gas supply from a source group (or well) may be augmented by advance gas defined in keyword GADVANCE (or WADVANCE). Gas used for fuel (GRUPFUEL) and sales (GRUPSALE) is by
default subtracted from a source group’s available gas supply for reinjection, but keyword WTAKEGAS may be used to alter the priorities for reinjection, fuel and sales.

4. The mnemonic for a wellstream to be used as make-up gas.

This is only required if make-up gas is to be used to make up the injection target of the group named in item 1, if insufficient injection gas is available from the source defined in items 2 and 3.

If make-up gas is used, the injection calculation may have to be iterated to match the injection gas composition with the injection rate (see keyword GCONTOL).

5. The stage of the separator which defines the fluid composition for injection.

The vapor from any stage of the separator may act as a source of injection fluid. The default value of 0 results in the use of the vapor from the whole separator as the injection fluid.

Note that the choice of separator stage also affects the available produced gas supply rate from the source group or well (item 3) if the source is GV or W (item 2).

DEFAULT: 0

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also keyword WAVAILIM, and items 3 and 6 of keyword GCONINJE, which can be used to control the rate of gas injection.

Notes

For accurate group gas accounting, there is a limitation that re-injection control and any sales gas or fuel gas requirements must all be specified for the same group, usually the FIELD group, and not split across the group hierarchy. This limitation only applies if the injection gas contains group vapor, with the vapor taken from a separator stage (that is not from the whole separator).

Example

The gas injection groups are to use the vapor produced by the FIELD group.

```plaintext
GINJGAS
'GI*' GV FIELD /
/
```

Keywords

942
Gi nodal values

The data comprises NTPVT (item 2 in keyword \texttt{TABDIMS}) tables of Gi values, each terminated by a slash (/), describing the nodal values for Gi to be used in the calculation of the oil and gas properties. Each table consists of up to NPPVT real entries.

The Gi nodal values within each table should be positive and strictly increasing.

UNITs: $\text{sm}^3/\text{rm}^3$ (METRIC), Mscf/rb (FIELD), scc/rcc (LAB)

The number of entries in each table must correspond to the number of multiplier values entered under each of the \texttt{RSGI}, \texttt{RVGI}, \texttt{BGGI} and \texttt{BOGI} keywords.

See "Gi Pseudo-compositional model" in the \textit{ECLIPSE Technical Description} for further information.

Example

With NTPVT=2 and NPPVT $\geq$ 4

<table>
<thead>
<tr>
<th>GINODE</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
</tr>
<tr>
<td>10.0</td>
</tr>
<tr>
<td>50.0</td>
</tr>
<tr>
<td>100.0 /</td>
</tr>
<tr>
<td>0.0</td>
</tr>
<tr>
<td>100.0 /</td>
</tr>
</tbody>
</table>

Keywords
Maximum group capacity for artificial lift

The keyword can be used to limit the number of wells that are automatically switched to artificial lift (see keyword `WLIFT` and `PRORDER`). Limits can be set for individual groups or the field as a whole. A well will not be switched to artificial lift if it would exceed the lift capacity of its superior groups (including the field). Lift capacity limits will not prevent retubing from being carried out (see item 4 in keyword `WLIFT`). No action will be taken if wells already placed on artificial lift exceed these limits.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

1. Group name or group name root
   or FIELD (if the limits apply to the field as a whole)

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Maximum total lift capacity.

   This acts as a limit on the sum of the artificial lift quantity (ALQ) values of all subordinate open producers multiplied by their efficiency factors. If the ALQ refers to the pump power, this item limits the total pump power that can be applied in the group. For gas lift, if the ALQ refers to the lift gas injection rate, this item limits the group's total lift gas injection rate.

   Note: If you wish to omit the efficiency factors from the sum of the well ALQ values, set item 49 in OPTIONS to > 0. This restores back-compatibility with pre-96A versions of ECLIPSE, in which efficiency factors are not taken into account. Alternatively, group efficiency factors could be applied instead (see keyword `GEFAC`).

   If the item is defaulted or set to zero, this limit will not be applied.

   UNITS: These depend on what the ALQ was meant to represent when the table was created. Refer to section "Record 6:ALQ values" in the description of keyword `VFPPROD`.

3. Maximum number of wells on artificial lift. This acts as a limit on the number of subordinate open producers that have a non-zero value for their artificial lift quantity.

   If the item is defaulted or set to zero, this limit is not applied.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Example

```
GLIFTLIM
PLAT-A  800.0    /
PLAT-B  1*      20 /
/            
```

Keywords

944
GLIFTOPT

Group lift gas limits for gas lift optimization

The GLIFTOPT keyword may be used to set group lift gas supply limits for the Gas lift optimization facility (see "Gas lift optimization" in the ECLIPSE Technical Description). Keyword LIFTOPT must be entered before GLIFTOPT, to activate the facility.

The keyword is followed by any number of records, each containing the following items of data and terminated with a slash (/). The set of records must end with a blank record, containing only a slash.

1. Group name or group name root, or FIELDS to set a lift gas supply limit for the field
   
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Maximum lift gas supply limit for the group
   
   The group’s lift gas supply is the sum of the lift gas injection rates of its subordinate wells or groups, multiplied by the wells’ or groups’ efficiency factors.

   A default or negative value implies there is no limit for the group.

   UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

3. Maximum total gas rate for the group
   
   The group’s total gas rate is the sum of the lift gas plus the gas produced from the formation for each subordinate well or group, multiplied by the well’s or group’s efficiency factor. Lift gas increments are not allocated to wells below this group if they would cause this limit to be exceeded.

   A default or negative value implies there is no limit for the group.

   UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

End the data record with a slash (/).

End the set of records with a blank record containing just a slash (/).

**Example**

<table>
<thead>
<tr>
<th>GLIFTOPT</th>
</tr>
</thead>
<tbody>
<tr>
<td>-- group max lift max total</td>
</tr>
<tr>
<td>-- name gas rate gas rate</td>
</tr>
<tr>
<td>-- SM$^3$/DAY</td>
</tr>
<tr>
<td>'PL-*' 1e6 /</td>
</tr>
</tbody>
</table>

---

**Keywords**

ECLIPSE Reference Manual

945
GLTHEX

**Geomechanical linear thermal expansion coefficient**

This keyword may only be used when the `GEOMECH` keyword is specified in the RUNSPEC section together with one of `THERMAL`, `TEMPI` or `TEMPVD`.

The keyword should be followed by one non-negative real number for every grid block in the current input box, specifying the rock linear thermal expansion coefficient. The data must be terminated by a slash (/).

**UNITS:** 1/K (`METRIC`), 1/°R (`FIELD`), 1/K (`LAB`), 1/K (`PVT-M`)

An additional term is included in the elastic stiffness matrix, \((2G + 3\lambda)\alpha_T(T - T_r)\) if the THERMAL option is used or \((2G + 3\lambda)\alpha_T T\) if the THERMAL option is not used. In these relations, \(G\) is the modulus of rigidity, \(\lambda\) is the Lame constant, \(\alpha_T\) is the coefficient of linear expansion defined by this keyword and \(T_r\) is a reference temperature. Temperature \(T\) in the above calculation is predicted when using the THERMAL option and is set with the keywords `TEMPI` or `TEMPVD` if not.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y- and Z-axis indices. Repeat counts may be used for repeated values (for example 115*1.25E-5). Note that spaces may not be inserted on either side of the asterisk.

**Example**

```
GLTHEX 200*1.25E-5 /
```
GMDISBC  Displacement boundary conditions for coupled rock and stress calculations

This keyword specifies displacement boundary conditions when a coupled rock stress and fluid flow calculation is specified with the keyword GEOMECH in the RUNSPEC section.

This keyword allows displacement boundary conditions to be specified on all boundaries. If a particular boundary is not specified by this keyword or by GMTRABC or GMPSTBC keywords, that boundary is defaulted to a rigid boundary.

This keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the displacement boundary condition is applied
   This must be one of the following:
   - X+  Specify the boundary condition on the outer X+ edge
   - X-  Specify the boundary condition on the outer X- edge
   - Y+  Specify the boundary condition on the outer Y+ edge
   - Y-  Specify the boundary condition on the outer Y- edge
   - Z+  Specify the boundary condition on the outer Z+ edge
   - Z-  Specify the boundary condition on the outer Z- edge

   The Z- boundary is the top boundary of the simulation, Z+ is the bottom.

2. An integer specifying the type of boundary condition on this face
   This must be one of the following:
   1. A normal rock displacement is specified on the boundary denoted in item 1.
   2. A rock displacement with X, Y, Z components. A single X, Y, Z displacement component is set in item 3 and the component name must be set in item 4. Separate lines are needed to set different components.

   For boundary condition type 2, the X, Y, Z components conform to the X, Y, Z directions as specified in the COORD keyword for corner point geometry.

   DEFAULT: 2

3. A normal rock displacement on this boundary if the boundary condition type in item 2 is 1, or an X, Y, Z component of a rock displacement if the boundary condition type in item 2 is 2.

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
4. A character string that gives the component of displacement if the boundary type in item 2 is 2. This must be one of ‘X’, ‘Y’ or ‘Z’.
   DEFAULT: None. It must be specified if item 2 is 2
5. First block to be modified in the X direction
   DEFAULT: 1 if an X- face
6. Last block to be modified in the X direction
   DEFAULT: NX if an X+ face
7. First block to be modified in the Y direction
   DEFAULT: 1 if a Y- face
8. Last block to be modified in the Y direction
   DEFAULT: NY if a Y+ face
9. First block to be modified in the Z direction
   DEFAULT: 1 if a Z- face
10. Last block to be modified in the Z direction
    DEFAULT: NZ if a Z+ face

Example

```plaintext
-- set a normally rigid boundary in the x and y-directions
GMDISBC
-- side type displacement
  'X+' 1 /
  'X-' 1 /
  'Y+' 1 /
  'Y-' 1 /
/
```

Keywords

948
This keyword specifies displacement boundary conditions when a coupled rock stress and fluid flow calculation is specified with the keyword GEOMECH in the RUNSPEC section.

Use of this keyword allows displacement boundary conditions to be specified on all boundaries of a local grid refinement. If a particular boundary is not specified in this keyword or the GMTRABCL or GMFSTBCL keywords, that boundary is defaulted to a rigid boundary.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the velocity boundary condition is applied
   This must be one of the following:
   
   X+ Specify the boundary condition on the outer X+ edge
   X- Specify the boundary condition on the outer X- edge
   Y+ Specify the boundary condition on the outer Y+ edge
   Y- Specify the boundary condition on the outer Y- edge
   Z+ Specify the boundary condition on the outer Z+ edge
   Z- Specify the boundary condition on the outer Z- edge

   DEFAULT: There is no default for this item; it must be specified.

   The Z- boundary is the top boundary of the simulation, Z+ is the bottom.

2. An integer specifying the type of boundary condition on this face
   This must be one of the following:
   a. A normal rock displacement is specified on the boundary denoted in item 1.
   b. A rock displacement with X, Y, Z components.

      A single X, Y, Z component is set in item 4 and the component name must be set in item 5.
      Separate lines are needed to set different components.

      For boundary condition type 2, the X, Y, Z components conform to the X, Y, Z directions as specified in the COORD keyword for corner point geometry.

      DEFAULT 2

3. Name of the Local Grid Refinement
4. A normal rock displacement on this boundary, if the boundary condition type in item 2 is 1, or an X, Y, Z component of a rock displacement, if the boundary condition type in item 2 is 2.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: 0

5. A character string that gives the component of displacement or velocity if the boundary type in item 2 is 2. This must be one of ‘X’, ‘Y’ or ‘Z’.

DEFAULT: None. It must be specified if item 2 is 2

6. First block to be modified in the X direction in the LGR box specified in item 3.

DEFAULT: 1 if an X- face

7. Last block to be modified in the X direction in the LGR box specified in item 3.

DEFAULT: NX if an X+ face

8. First block to be modified in the Y direction in the LGR box specified in item 3.

DEFAULT: 1 if a Y- face

9. Last block to be modified in the Y direction in the LGR box specified in item 3.

DEFAULT: NY if a Y+ face

10. First block to be modified in the Z direction in the LGR box specified in item 3.

DEFAULT: 1 if a Z- face

11. Last block to be modified in the Z direction in the LGR box specified in item 3.

DEFAULT: NZ if a Z+ face

Example

```plaintext
-- set a normally rigid boundary in the x and y-
directions for the local grid refinement LGR1
GMDISBC
-- side type velocity
   'X+'  1   'LGR1'  /
   'X-'  1   'LGR1'  /
   'Y+'  1   'LGR1'  /
   'Y-'  1   'LGR1'  /
   /
```

Keywords

950
GMPSTBC  Principal stress boundary conditions coupled stress and fluid flow

This keyword specifies principal stress boundary conditions when a finite volume or finite element rock stress calculation is specified. It is only applicable when the keyword GEOMECH has been specified in the RUNSPEC section.

This keyword allows the full set of three principal stress vectors to be specified on all boundaries. This is sufficient information to fully define the stress tensor at the chosen boundary. Alternatively, tractions or displacements may be specified using the GMRABC or GMDISBC keywords.

Note that the principal stress vectors should, by definition, be mutually orthogonal. If non-orthogonal vectors are provided, ECLIPSE 300 issues a warning message and orthogonalizes the vectors provided using the Gram-Schmidt process.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the principal stress boundary condition is applied
   This must be one of the following:
   X+  Specify the boundary condition on the outer X+ edge
   X−  Specify the boundary condition on the outer X- edge
   Y+  Specify the boundary condition on the outer Y+ edge
   Y−  Specify the boundary condition on the outer Y- edge
   Z+  Specify the boundary condition on the outer Z+ edge
   Z−  Specify the boundary condition on the outer Z- edge

   The Z- boundary is the top boundary of the simulation, Z+ is the bottom.

2. The X component of the first principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

3. The Y component of the first principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

4. The Z component of the first principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
5. The X component of the second principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

6. The Y component of the second principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

7. The Z component of the second principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

8. The X component of the third principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

10. The Z component of the third principal stress vector.
    UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
     DEFAULT: 0

11. First block to be modified in the X direction
    DEFAULT: 1 if an X- face

12. Last block to be modified in the X direction
    DEFAULT: NX if an X+ face

13. First block to be modified in the Y direction
    DEFAULT: 1 if a Y- face

14. Last block to be modified in the Y direction
    DEFAULT: NY if a Y+ face

15. First block to be modified in the Z direction
    DEFAULT: 1 if a Z- face

16. Last block to be modified in the Z direction
    DEFAULT: NZ if a Z+ face
Example

-- Principal stress vectors:
-- P1=(-2000,0,0), P2=(0,-2500,200), P3=(0,0,0)
-- are defined on all boundary faces of the entire grid.

<table>
<thead>
<tr>
<th>side</th>
<th>P1X</th>
<th>P1Y</th>
<th>P1Z</th>
<th>P2X</th>
<th>P2Y</th>
<th>P2Z</th>
<th>P3X</th>
<th>P3Y</th>
<th>P3Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>'X-'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'X+'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'Y-'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'Y+'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'Z-'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'Z+'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
</tbody>
</table>
GMPSTBCL  Principal stress boundary conditions for coupled stress and fluid flow in a Local Grid Refinement

This keyword specifies principal stress boundary conditions when a finite volume or finite element rock stress calculation is specified. It is only applicable when the keyword `GEOMECH` has been specified in the `RUNSPEC` section.

This keyword allows the full set of three principal stress vectors to be specified on all boundaries in a local grid refinement. This is sufficient information to fully define the stress tensor at the chosen boundary. Alternatively, tractions or displacements may be specified using the `GMTRABCL` or `GMDISBCL` keywords.

Note that the principal stress vectors should, by definition, be mutually orthogonal. If non-orthogonal vectors are provided, ECLIPSE 300 issues a warning message and orthogonalizes the vectors provided using the Gram-Schmidt process.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the principal stress boundary condition is applied

   This must be one of the following:

   X+  Specify the boundary condition on the outer X+ edge
   X−  Specify the boundary condition on the outer X- edge
   Y+  Specify the boundary condition on the outer Y+ edge
   Y−  Specify the boundary condition on the outer Y- edge
   Z+  Specify the boundary condition on the outer Z+ edge
   Z−  Specify the boundary condition on the outer Z- edge

   DEFAULT: There is no default for this item; it must be specified.

   The Z- boundary is the top boundary of the simulation, Z+ is the bottom.

2. Name of local grid refinement

3. The X component of the first principal stress vector.

   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

   DEFAULT: 0

4. The Y component of the first principal stress vector.

   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

   DEFAULT: 0
5. The Z component of the first principal stress vector.
   
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   
   DEFAULT: 0

6. The X component of the second principal stress vector.
   
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   
   DEFAULT: 0

7. The Y component of the second principal stress vector.
   
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   
   DEFAULT: 0

8. The Z component of the second principal stress vector.
   
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   
   DEFAULT: 0

9. The X component of the third principal stress vector.
   
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   
   DEFAULT: 0

10. The Y component of the third principal stress vector.
    
    UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
    
    DEFAULT: 0

11. The Z component of the third principal stress vector.
    
    UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
    
    DEFAULT: 0

12. First block to be modified in the X direction
    
    DEFAULT: 1 if an X- face

13. Last block to be modified in the X direction
    
    DEFAULT: NX if an X+ face

14. First block to be modified in the Y direction
    
    DEFAULT: 1 if a Y- face

15. Last block to be modified in the Y direction
    
    DEFAULT: NY if a Y+ face

16. First block to be modified in the Z direction
    
    DEFAULT: 1 if a Z- face

17. Last block to be modified in the Z direction
    
    DEFAULT: NZ if a Z+ face
Example

-- Principal stress vectors:
-- P1=(-2000,0,0), P2=(0,-2500,200), P3=(0,0,0)
-- are defined on all boundary faces of the local grid refinement LG1.

<table>
<thead>
<tr>
<th>side</th>
<th>LGR</th>
<th>P1X</th>
<th>P1Y</th>
<th>P1Z</th>
<th>P2X</th>
<th>P2Y</th>
<th>P2Z</th>
<th>P3X</th>
<th>P3Y</th>
<th>P3Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>'X-'</td>
<td>'LG1'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td></td>
<td></td>
</tr>
<tr>
<td>'X+'</td>
<td>'LG1'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td></td>
<td></td>
</tr>
<tr>
<td>'Y-'</td>
<td>'LG1'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td></td>
<td></td>
</tr>
<tr>
<td>'Y+'</td>
<td>'LG1'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td></td>
<td></td>
</tr>
<tr>
<td>'Z-'</td>
<td>'LG1'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td></td>
<td></td>
</tr>
<tr>
<td>'Z+'</td>
<td>'LG1'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

/
GMTRABC  Traction boundary conditions

This keyword allows traction boundary conditions to be specified on all boundaries. A traction at a boundary surface is a force/unit area. If a particular boundary is not specified, that boundary is defaulted to a rigid boundary unless a displacement condition is given with the keyword GMDISBC or a principal stress boundary condition is given with keyword GMPSTBC.

This keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the traction boundary condition is applied
   This must be one of the following:
   
   X+  Specify the boundary condition on the outer X+ edge
   X−  Specify the boundary condition on the outer X- edge
   Y+  Specify the boundary condition on the outer Y+ edge
   Y−  Specify the boundary condition on the outer Y- edge
   Z+  Specify the boundary condition on the outer Z+ edge
   Z−  Specify the boundary condition on the outer Z- edge

   DEFAULT: There is no default for this item; it must be specified.

   The Z- boundary is the top boundary of the simulation, Z+ is the bottom.

2. An integer specifying the type of boundary condition on this face
   This must be one of the following:
   
   1  A total normal traction is specified on the boundary denoted in item 1. The magnitude of this normal traction is set in item 3.

   2  A given traction with X, Y, Z components. A single X, Y, Z traction component is set in item 3 and the component name is set in item 4. Separate lines are needed to set different components.

   DEFAULT: If a boundary of a cell or face is not specified with traction, stress or displacement boundary conditions, default is a rigid boundary as specified in GMDISBC item 2.

3. A normal traction on this boundary, if the boundary condition type in item 2 is 1, or an X or Y or Z component of a given traction, if the boundary condition type in item 2 is 2. The component direction is set in item 4 below.

   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

   DEFAULT:0
4. Finite element: A character string that gives the component of traction if the boundary type in item 2 is 2. This must be one of ‘X’, ‘Y’ or ‘Z’.
   DEFAULT: None. It must be specified if item 2 is 2

5. First block to be modified in the X direction
   DEFAULT: 1 if an X- face

6. Last block to be modified in the X direction
   DEFAULT: NX if an X+ face

7. First block to be modified in the Y direction
   DEFAULT: 1 if a Y- face

8. Last block to be modified in the Y direction
   DEFAULT: NY if a Y+ face

9. First block to be modified in the Z direction
   DEFAULT: 1 if a Z- face

10. Last block to be modified in the Z direction
    DEFAULT: NZ if a Z+ face

Example

```
-- set a 3000 psi total compressive traction on the X+, X-, Y+ and Y- boundaries
-- set a 3000 psi overburden pressure (compressive stress is -ve)
-- note that the ‘Z-’ direction is positive downwards
GMTRABC
-- side type total
-- traction
  'X+'  1   -3000 /
  'X-'  1   -3000 /
  'Y+'  1   -3000 /
  'Y-'  1   -3000 /
  'Z-'  1   -3000 /
/
```

Keywords

958
GMTRABCL  Traction boundary conditions in a Local Grid Refinement

This keyword specifies traction boundary conditions when a coupled stress calculation is specified with the keyword GEOMECH in the RUNSPEC section.

This keyword allows traction boundary conditions to be specified on all boundaries of a local grid refinement. A traction at a boundary surface is a force/unit area. If not specified, that boundary is defaulted to a rigid boundary unless a displacement condition is given with the keyword GMDISBC or a principal stress boundary condition is given with keyword GMPSTBCL.

This keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the traction boundary condition is applied
   This must be one of the following:
   - X+  Specify the boundary condition on the outer X+ edge
   - X-  Specify the boundary condition on the outer X- edge
   - Y+  Specify the boundary condition on the outer Y+ edge
   - Y-  Specify the boundary condition on the outer Y- edge
   - Z+  Specify the boundary condition on the outer Z+ edge
   - Z-  Specify the boundary condition on the outer Z- edge
   DEFAULT: There is no default for this item; it must be specified.
   The Z- boundary is the top boundary of the simulation, Z+ is the bottom.

2. An integer specifying the type of boundary condition on this face
   This must be one of the following:
   a. A total normal traction is specified on the boundary denoted in item 1. The magnitude of this normal traction is set in item 4.
   b. A given traction with X, Y, Z components. A single X, Y, Z traction component is set in item 4 and the component name is set in item 5. Separate lines are needed to set different components.
   DEFAULT: If a boundary of a cell or face is not specified with traction, stress or displacement boundary conditions, default is a rigid boundary as specified in GMDISBC item 2.

3. Name of the Local Grid Refinement

4. A normal traction on this boundary, if the boundary condition type in item 2 is 1, or an X or Y or Z component of a given traction, if the boundary condition type in item 2 is 2. The component direction is set in item 5 below.
UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

DEFAULT: 0

5. A character string that gives the component of traction if the boundary type in item 2 is 2. This must
   be one of ‘X’, ‘Y’ or ‘Z’.
   DEFAULT: None. It must be specified if item 2 is 2

6. First block to be modified in the X direction
   DEFAULT: 1 if an X- face

7. Last block to be modified in the X direction
   DEFAULT: NX if an X+ face

8. First block to be modified in the Y direction
   DEFAULT: 1 if a Y- face

9. Last block to be modified in the Y direction
   DEFAULT: NY if a Y+ face

10. First block to be modified in the Z direction
    DEFAULT: 1 if a Z- face

11. Last block to be modified in the Z direction
    DEFAULT: NZ if a Z+ face

Example

```
-- For the local grid refinement LGR1 set a 3000 psi total compressive traction on the X +, X-, Y+ and Y- boundaries
-- set a 3000 psi overburden pressure (compressive stress is -ve)
-- note that the 'Z-' direction is positive downwards
GMTRABCL
  -- side type total
  -- traction
    'X+'  1  'LGR1'   -3000 /
    'X-'  1  'LGR1'   -3000 /
    'Y+'  1  'LGR1'   -3000 /
    'Y-'  1  'LGR1'   -3000 /
    'Z-'  1  'LGR1'   -3000 /
/```

Keywords

960
GNETDP

Controls the dynamic changing of fixed pressures

This keyword may be used to adjust the pressure of a fixed-pressure group or node in the network option (see "Network option" in the ECLIPSE Technical Description), with the aim of maintaining its production flow rate within specified maximum and minimum limits. Whenever the flow into the fixed-pressure group or node falls below the minimum rate specified in item 3, ECLIPSE adjusts its pressure value by subtracting a pressure increment specified in item 5, then rebalances the network. This process continues until either the flow exceeds the minimum value or the pressure falls to the minimum allowed pressure specified in item 7. Similarly, whenever the flow into the fixed-pressure group or node rises above the maximum rate specified in item 4, ECLIPSE adjusts its pressure value by adding a pressure increment specified in item 6, then rebalances the network. This process continues until either the flow falls below the maximum value or the pressure increases to the maximum allowed pressure specified in item 8.

The facility is available with both the Standard Network Model (for fixed-pressure groups) and the Extended Network Model (for fixed-pressure nodes). Subordinate groups or nodes cannot have their pressure adjusted with this facility.

This facility predates automatic chokes (see "Automatic chokes" in the ECLIPSE Technical Description), which offer a more versatile means of controlling a group’s flow rate by adjusting the network nodal pressures. However, automatic chokes are available only in the Extended Network Model (see "The extended network model" in the ECLIPSE Technical Description).

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Name of the fixed-pressure group, for a standard production network,
   or
   name of the fixed-pressure node, for an extended network

2. Phase whose rates items 3 and 4 refer to
   OIL, WAT, GAS, or LIQ
   DEFAULT: GA

3. Minimum rate (into the group/node of item 1) that triggers a pressure adjustment when violated.
   UNITS: sm³/day (METRIC), stb/day for oil, water or liquid (FIELD), Mscf/day for gas (FIELD), scc/hr (LAB), sm³/day (PVT-M)
   DEFAULT: 0.0 (there will be no pressure adjustment to increase the flow)

4. Maximum rate (into the group/node of item 1) that triggers a pressure adjustment when violated.
   UNITS: sm³/day (METRIC), stb/day for oil, water or liquid (FIELD), Mscf/day for gas (FIELD), scc/hr (LAB), sm³/day (PVT-M)
   DEFAULT: infinity (1.0E20) (there will be no pressure adjustment to decrease the flow)

5. Pressure increment to be subtracted from the fixed pressure when the flow rate falls below the minimum in item 3.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0.0

6. Pressure increment to be added to the fixed pressure when the flow rate exceeds the maximum in item 4.
UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
DEFAULT: 0.0

7. Minimum allowable pressure at the group/node.
The pressure is not adjusted below this value.
UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
DEFAULT: 0.0

8. Maximum allowable pressure at the group/node.
The pressure is not adjusted above this value.
UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
DEFAULT: Infinity (1.0E20)

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Note: The network has to be rebalanced each time the pressure is adjusted so that the flows may be recalculated at the new pressure. The smaller the increment sizes are in items 5 and 6, the number of times the pressure is adjusted is correspondingly greater, increasing the computing time.
If both minimum and maximum rate limits are set in items 3 and 4, the pressure increments must be made small enough to find pressure values at which the rate lies between these limits. The minimum and maximum limits must not lie too close together.

Example

```
GNETDP
-- Node    Phase    Min      Max      DP     DP     Min    Max
-- Name             Rate     Rate    Subt    Add    Pres   Pres
FIELD    GAS     5000.0   15000.0  10.0    10.0   300.0  600.0 /
/
```

In this example the fixed pressure group/node FIELD has its fixed pressure changed according to how much gas is flowing into it. If the rate falls below 5000 then the fixed pressure is reduced in increments of 10 psi until the constraint is satisfied. If the rate goes above 15000 then the fixed pressure is raised in increments of 10 psi until the constraint is satisfied. The fixed pressure is not allowed to fall below 300 psi, or rise above 600 psi, regardless of the rates into the group/node.
GNETINJE  Defines the structure of the injection network

The keyword defines the structure of an injection network in the Network option (see "Network option" in the ECLIPSE Technical Description). Injection networks exist independently of the production network (if present), and separate networks can be defined for water and gas injection.

In ECLIPSE 300, if you are using the Extended Network keywords (NODEPROP and BRANPROP) to define the production network, you must also use those keywords instead of GNETINJE to define any injection networks; extended and standard network models cannot be mixed. In ECLIPSE 100, you must use GNETINJE to define any injection networks irrespective of the model used for the production network.

The keyword is followed by any number of records, each containing the following items of data and terminated with a slash (/). The set of records must end with a blank record, containing only a slash.

1. Group name or group name root,
   or **FIELD** (to set a fixed pressure for the field node)
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.
2. Phase of the injection network in which this group is a node
   **WAT** or **GAS**
3. Fixed pressure for the group corresponding to the terminal node of the network
   Default this item, or set it negative, if the group is not a fixed-pressure terminal node in the network.
   **UNITS:** barsa (**METRIC**), psia (**FIELD**), atma (**LAB**), atma (**PVT-M**)
4. Injection VFP table number for the pipeline between the group and its parent group
   A value of 9999 can be entered to indicate that there is no pressure loss in the network branch between the group and its parent group. Otherwise, if a non-zero table number is entered, the corresponding injection VFP table must have been set up for the phase named in item 2.
   If the group is a fixed-pressure node (that is if a positive value is set for item 3) this item must be defaulted, since a fixed-pressure group is a terminal node and cannot have a pipeline connection to a higher group.
   If this item is defaulted for a group that is not a fixed-pressure node, the group is not part of the network.
   Two or more network pipelines may share the same VFP table if they have similar pressure loss characteristics.
   **DEFAULT:** 0 Implies the group has no pipeline connection to its parent group.

End the data record with a slash (/).
End the set of records with a blank record containing just a slash.

**Example**

```
GNETINJE
PLAT-A  WAT  14.7  /
GR-B2   WAT  1*   1  /
/
```
Automatic pump/compressor switching in a standard network

The keyword defines automatic compressors or pumps in a standard production network in the network option (see "Network option" in the ECLIPSE Technical Description). If you are using the extended network model (keyword \textsc{NETWORK} in the \textsc{RUNSPEC} section), you should define automatic compressors or pumps with keyword \textsc{NETCOMPA} instead.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated at any item after item 1. The remaining items assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Production rate below which the pump/compressor is to be switched on
   \textsc{UNITS}: \textsc{sm}^{3}/\textsc{day} (\textsc{METRIC}), \textsc{stb}/\textsc{day} for oil (\textsc{FIELD}), \textsc{Mscf}/\textsc{day} for gas (\textsc{FIELD}), \textsc{scc}/\textsc{hr} (\textsc{LAB}), \textsc{sm}^{3}/\textsc{day} (\textsc{PVT-M})
   \textsc{DEFAULT}: 0.0
   A zero or negative value will disable this option.

3. Phase to which the production rate in item 2 refers
   \textsc{OIL} or \textsc{GAS}
   \textsc{DEFAULT}: \textsc{OIL}

4. New VFP table number
   (When the pump/compressor is switched on this replaces the table number previously specified in keyword \textsc{GRUPNET}.)
   A zero or negative number will leave the original VFP table number unchanged.
   \textsc{DEFAULT}: 0

5. New artificial lift quantity (ALQ)
   (When the pump/compressor is switched on this replaces the artificial lift quantity previously specified in keyword \textsc{GRUPNET}.)
   The artificial lift quantity may be regarded as the pump or compressor power, according to the definition used in the VFP table.
   \textsc{UNITS}: These depend on what the ALQ was meant to represent when the table was created. Refer to section "Record 6:ALQ values" in the description of keyword \textsc{VFPPROD}.
   \textsc{DEFAULT}: 0.0

6. New gas consumption rate for the group
(When the pump/compressor is switched on this replaces any consumption rate previously specified with keyword GCONSUMP, to account for the increase in gas demand for power generation.)

A zero or negative value leaves the original gas consumption rate unchanged (that is, zero if GCONSUMP is not specified).

The gas consumption is subtracted from the flow rate through the pipeline when interpolating the VFP table to calculate the pressure loss. Gas consumption is also taken into account for group reinjection control (see keyword GCONINJE) and sales gas control (see keyword GCONSALE). But the total gas production rates reported for the groups and the field will not be reduced.

UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT–M)

DEFAULT: 0.0

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

The pump/compressor is switched on at the end of the timestep in which the group’s production rate falls below the set limit. A network balancing calculation is performed at the beginning of the next timestep, regardless of the interval set in keyword NETBALAN. However, if a limit tolerance fraction has been set with keyword WLIMTOL, the timestep in which the rate fell below the limit is recalculated with the pump/compressor switched on if the limit is violated by more than this fraction.

**Note:** An automatic compressor cannot be present if the pipeline’s ALQ is equated to the oil or gas surface density in item 7 of keyword GRUPNET.

---

**Example**

Pumps to boost the flow through the network pipelines from platforms PLAT-A and PLAT-B are started if their oil production rates fall below 1000 stb/day. An artificial lift quantity of 100 is used to interpolate the VFP table that calculates the pressure loss in the pipeline from PLAT-A, keeping the same VFP table number. The pipeline from PLAT-B changes to a new table number 3, interpolated with a zero artificial lift quantity. There is no change to any gas consumption rate, if specified.

```
GNETPUMP
 PLAT-A  1000  OIL  0 100 /
 PLAT-B  1000  OIL  3 /
```

---

**Keywords**

965
Data for pressure maintenance groups

This keyword may be used to instruct a group to adjust its production or injection rate so as to maintain the average pressure in a particular fluid-in-place region at a specified target.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root
   or FIELD (to adjust the field’s production or injection rate)

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

   The specified group maintains the average pressure in the selected region, by adjusting its production or injection rate as chosen in item 2. Obviously the group’s wells must be able to influence the region’s pressure, preferably being completed inside the region. It is possible to specify more than one group to maintain the pressure in a particular region. The groups’ flow rate targets are set in proportion to the proportionality constant in item 6.

2. Flow target that ECLIPSE calculates to control the pressure

   PROD The group’s production RESV rate is adjusted.

   ECLIPSE 100 only
     OINJ The group’s oil injection RESV rate is adjusted.
     WINJ The group’s water injection RESV rate is adjusted.
     GINJ The group’s gas injection RESV rate is adjusted.

   ECLIPSE 100 only
     OINS The group’s oil injection surface rate is adjusted.
     WINS The group’s water injection surface rate is adjusted.
     GINS The group’s gas injection surface rate is adjusted.

   ECLIPSE 300 only
     WGIN The group’s wet gas injection rate is adjusted.

   NONE The group’s flow is no longer adjusted to maintain pressure.

   If PROD is selected, the group must not be under prioritization group control (see keyword GCONPRI), as only guide rate group control (keyword GCONPROD) can be used to control the pressure.

   Note: When running ECLIPSE 300 in compositional mode, if one of the gas flow targets (that is, GINJ, GINS or WGIN) are selected in this item, then the nature of the injected gas must also be specified with keyword GINJGAS, unless the group is under control of a higher level group with gas injection controls where the injection fluid is defined.
**CAUTION:** Additional injection or production rate limits may be set for the group with keyword `GCONINJE` or `GCONPROD`. These limits are not removed by using the `GPMAINT` keyword, except for the limit that corresponds to the control target that is adjusted to maintain the pressure, which is overwritten at each timestep by the target calculated for pressure maintenance.

If the flow quantity selected in item 2 is at a later date no longer required to be adjusted for pressure maintenance, its rate target must be reset or removed with an appropriate group control keyword, otherwise the latest calculated target is retained as a limit.

### 3. Region number, whose pressure this group controls.

This **fluid-in-place** region should be identified by an integer between 0 and `NTFIP` (see keyword `REGDIMS` in the `RUNSPEC` section). Region 0 refers to the whole field. Fluid-in-place regions are defined with the keyword `FIPNUM` in the `REGIONS` section (or keyword `FIP` if a different family of regions is requested in item 4).

### 4. The fluid-in-place region family, to which the specified region belongs.

The family name should be up to 5 characters long.

A default (1*) or blank (" ") entry refers to the standard set of fluid-in-place regions defined with the keyword `FIPNUM`.

Additional families of fluid-in-place regions may optionally be defined, using the keyword `FIP`. The family name is string of up to 5 characters that makes up the second segment of the keyword name (characters 4 - 8). Each family can divide up the field in different ways. Thus, for example, the standard `FIPNUM` family may divide the field into layers, and an additional family can be defined to divide the field into areal sectors. Pressure maintenance may be selected for any region in any family.

### 5. The target pressure for the region

The group attempts to maintain the region’s average hydrocarbon pressure at this value.

**UNITS:** barsa (METRIC), psi (FIELD), atma (LAB), atma (PVT-M)

**Note:** The field pressure `FPRP` shown in the screen output and in the Print file is a pore-volume weighted pressure and may not correspond exactly to the hydrocarbon pore-volume weighted target pressure, `FPR`.

### 6. The proportionality constant, for controlling the group’s flow rate

At each timestep, the group’s target rate is set equal to the proportionality constant multiplied by the region’s pressure error term.

**UNITS:** m³ or sm³/day/bars (METRIC), rb or stb/day/psi (FIELD), rcc or scc/hr/atm (LAB), m³ or sm³/day/atm (PVT-M)

### 7. The integral time constant, for controlling the group’s flow rate

The integral of the region’s pressure error over time, divided by the integral time constant, is added to the current pressure error.
Example

<table>
<thead>
<tr>
<th>GPMAINT</th>
<th>Group</th>
<th>Inj</th>
<th>FIP</th>
<th>FIP</th>
<th>Targ</th>
<th>Prop</th>
<th>Inte</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>Phase</td>
<td>Reg</td>
<td>Fam</td>
<td>Pres</td>
<td>Const</td>
<td>Tcon</td>
<td></td>
</tr>
<tr>
<td>PLAT-A</td>
<td>WINJ</td>
<td>2</td>
<td>1*</td>
<td>3900</td>
<td>30</td>
<td>70</td>
<td></td>
</tr>
<tr>
<td>SP-C</td>
<td>WINJ</td>
<td>6</td>
<td>1*</td>
<td>3800</td>
<td>30</td>
<td>70</td>
<td></td>
</tr>
<tr>
<td>SP-B</td>
<td>PROD</td>
<td>3</td>
<td>1*</td>
<td>3600</td>
<td>50</td>
<td>70</td>
<td></td>
</tr>
</tbody>
</table>

Control strategy

The control strategy applied to the group’s flow target is that of a proportional + integral controller from elementary control theory. However, because of the discrete nature of the timesteps, the integral is replaced by a summation over the timesteps.

The group’s flow target for the quantity specified in item 2 (for example RESV) is set at each timestep equal to

\[
V = V_i + K\left[ e + \frac{\sum e \cdot \delta t}{T_i} \right] \tag{Eq. 3.76}
\]

where:

- \(V_i\) is the initial rate at the time the keyword is entered
- \(K\) is the proportionality constant (item 6)
- \(T_i\) is the integral time constant (item 7)
- \(e\) is the region’s pressure error \(P_{targ} - P_{reg}\) at the end of the previous timestep
- \(\sum(e \cdot \delta t)\) is the cumulative sum of the pressure error times the timestep length, up to the previous timestep, starting from the time the keyword was most recently entered.

The purpose of the \(V_i\) term is to help stabilize cases where a pressure maintenance target is set (or re-set with a subsequent GPMAINT keyword) when the flow rate of the controlled quantity is already adequate to maintain pressure. If the pressure error is small the flow should remain approximately at its initial value. But if pressure maintenance by injection is specified at the start of injection, \(V_i\) will be zero and the injection will build up from zero in response to the sum of the error and integral terms.

Increasing the proportionality constant \(K\) in item 6 helps the region’s pressure to reach its target value more quickly. But values of \(K\) that are too high result in oscillations in the pressure. A suitable order of magnitude to try initially for \(K\) would be the expected steady state flow rate (of the targeted quantity) of the group divided by a transient pressure error that you are prepared to tolerate.

Increasing the time constant \(T_i\) in item 7 helps to reduce the tendency for the pressure to oscillate, but will slow down the rate at which the pressure reaches its target value.

Each timestep acts as a “delay” in the control loop, as the pressure error is measured at the end of the previous timestep. The longer the timesteps are, the greater is the tendency for the pressure to oscillate for given values of \(K\) and \(T_i\). \(T_i\) should be set to a value at least as big as the maximum timestep size.
The resulting value of the flow target is applied as a target production or injection rate for the group. The group’s actual flow rate may still be limited by additional constraints set with keyword \texttt{GCONPROD} or \texttt{GCONINJE}. 

Keywords

969
**GPMAINT3 Data for pressure maintenance groups**

This keyword may be used to instruct a group to adjust its production or injection rate so as to maintain the average pressure in a particular pressure maintenance region at a specified target. It is retained for back compatibility with pre-2000A ECLIPSE 300 data files; however, you are encouraged to use keyword **GPMAINT** instead, which is compatible with both simulators.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records is ended by a blank record, containing only a slash.

1. **The name of the group.**
   
   A character string of up to 8 letters.

   This must be supplied, and a group of this name must exist.

   The specified group will maintain the average pressure in the selected region, by adjusting its production or injection rate as chosen in item 2. Obviously the group’s wells must be able to influence the region’s pressure, preferably being completed inside the region. It is possible to specify more than one group to maintain the pressure in a particular region. The groups’ flow rate targets are set in proportion to the proportionality constant in item 6.

2. **Flow target that ECLIPSE 300 calculates to control the pressure.**

   This may be one of the following:

   - **NONE** The group’s flow is no longer adjusted to maintain pressure
   - **PROD** The group’s production RESV rate is adjusted
   - **WINJ** The group’s water injection RESV rate is adjusted
   - **GINJ** The group’s gas injection RESV rate is adjusted (black oil mode only)
   - **WINS** The group’s water surface injection rate is adjusted
   - **GINS** The group’s gas surface injection rate is adjusted (black oil mode only)
   - **WAT** Water is injected, and the control target specified in item 9 is adjusted
   - **GAS** Gas is injected, and the control target specified in item 9 is adjusted. In a compositional run the composition of the injected gas is set to that of the field separator gas. However, it is generally better to specify the injected gas composition more precisely.
   - **ST** The molar composition of the injected fluid has been defined using the **WELLSTRE** keyword. The name of the injection stream must be specified in item 8. The control target specified in item 9 is adjusted.
   - **MIX** The molar composition of the injected fluid has been defined as a mixture using either the **WINMJMDX** or **WINJORD** keywords. The name of the mixture must be specified in item 8. The control target specified in item 9 is adjusted.
The molar composition of the injected fluid is to be taken from the vapor production of a nominated well. The name of the well must be specified in item 8. The control target specified in item 9 is adjusted.

The molar composition of the injected fluid is to be taken from the vapor production of a nominated group. The name of the group must be specified in item 8. The control target specified in item 9 is adjusted.

If PROD is selected, the group must not be under prioritization group control, as only guide rate group control can be used to control the pressure.

If any of WAT, GAS, ST, MIX, WV or GV are selected, the required pressure maintenance target rate will be assigned according to the specified control mode in item 9.

CAUTION: If there is a user-specified rate limit for the flow quantity selected here, it is overwritten by the target calculated for pressure maintenance. If the selected flow quantity is subsequently no longer required to be adjusted for pressure maintenance, its rate target must be reset with the appropriate group control keyword, otherwise the latest calculated target is retained.

3. Pressure maintenance region number, whose pressure this group controls.
   This should be an integer between 1 and NUMPMA (see keyword TABDIMS). The default value for the pressure maintenance region is 1.
   Pressure maintenance regions are defined with the PMANUM keyword.

4. This item is not used in ECLIPSE 300; it is retained for compatibility with ECLIPSE 100. The item must be defaulted.

5. The target pressure for the region
   The group attempts to maintain the region’s average hydrocarbon pore-volume weighted pressure at this value.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

Note: The field pressure FPRP in screen output and in the Print file is a pore-volume weighted pressure and may not correspond exactly to the hydrocarbon pore-volume weighted target pressure, FPR.

6. The proportionality constant, for controlling the group’s flow rate
   At each timestep, the group’s target rate is set equal to the proportionality constant multiplied by the region’s pressure error term.
   UNITS: rm³/day/bars (METRIC), rb/day/psi (FIELD), rcc/hr/atm (LAB), rm³/day/atm (PVT-M)

7. The integral time constant, for controlling the group’s flow rate
   The integral of the region’s pressure error over time, divided by the integral time constant, is added to the current pressure error.
   UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
8. A character string is required if the composition of the injection fluid was specified in item 2 using the ST, MIX, WV or GV methods.

   In these cases this item is a character string specifying the name of the wellstream, mixture, well or group that defines or supplies the injection stream.

9. A character string specifying the control target which is adjusted to maintain the pressure.

   This is required if item 2 was set to WAT, GAS, ST, MIX, WV or GV. It is one of the following:

   WA Water surface injection rate
   GA Gas surface injection rate
   RV Reservoir volume injection rate
   WG Wet gas rate

10. The maximum reinjection fraction

    This item provides an upper limit on the rate at which the group in item 1 may inject, with this rate limit defined as the reinjection fraction multiplied by the water or available gas rate of the group or well named in item 12. The available produced gas rate of a group is by default equal to the group’s gas production rate plus the rate of any advanced gas associated with the group (keyword GADVANCE) minus its fuel rate (keyword GRUPFUEL) minus its sales target (keyword GRUPSALE); however, this definition can be altered using keyword WTAKEGAS. In the case of a reinjection fluid defined by a stream or mixture, the specified fraction of the available surface gas production rate is applied as a wet gas reinjection rate limit.

    An alternative interpretation of this reinjection fraction may be activated by setting item 14 of this keyword to ‘PROD’, whereby the reinjection fraction limits the amount of available produced gas in the injection stream and not the overall rate of injection. This facility is only available when make-up gas has been specified to provide sufficient injection gas over and above the limited reinjection of produced gas for pressure maintenance (see item 13).

    DEFAULT: 0.0, giving no restriction on the reinjection fraction.

11. The type of production that the reinjection fraction multiplies.

    This need only be supplied if a reinjection fraction has been set in item 10. It should be one of the following:

    W The production rate of the well named in item 12
    G The production rate of the group named in item 12

    DEFAULT: W if item 2 is WV or GV otherwise

12. The name of the group or well that controls the reinjection limit.

    This need only be supplied if a reinjection fraction has been set in item 10. If item 11 is W it should be the name of a production well. If item 11 is G it should be the name of a group.

    If this name is not specified, it defaults to the fluid composition source name (item 8) for injection fluids WV or GV in item 2, and to the name of the group in item 1 for the other cases.

13. The mnemonic for a wellstream to be used as make-up gas.
This is only required if make-up gas is to be used when there is insufficient available gas from the source group or well to supply the injection gas for pressure maintenance.

14. This item defines the interpretation of the reinjection fraction defined in item 10 for gas injection:

- **TOT**: The gas injection rate of the group is limited by the reinjection fraction multiplied by the available produced gas rate of the group.

- **PROD**: When make-up gas has been defined (see item 13), the reinjection fraction limits the amount of produced gas used for injection, with any additional injection gas required for pressure maintenance being supplied from the make-up gas stream.

**DEFAULT**: TOT

**Notes**

Additional injection limits may be set for the group with the `GRUPINJE` keyword (or in the case where pressure is maintained by controlling the group’s production rate, additional production rate limits can be set with keyword `GRUPPROD`). These limits are not removed by using the `GPMAINT3` keyword, except for the limit that corresponds to the control target that is adjusted to maintain the pressure.

See also notes 2, 3 and 4 of keyword `GRUPINJE`.

**Examples**

**Example 1**

Basic pressure maintenance by water injection

```
GPMAINT3
-- Group Inj PMA Targ Prop Inte
-- Name Phase Reg Pres Const TCon
GR-B2 WINJ 0 1* 3500 100 30 /
```

**Example 2**

Pressure maintenance by gas injection, with only half the group’s produced gas used for this purpose. The make-up stream MUGAS provides an additional source of gas for injection should reinjection of the limited produced gas be insufficient to maintain the target pressure.

```
GPMAINT3
-- Group Inj PMA Targ Prop Time Srce Cont Reinj M-U Re-Inj
-- Name Fld Reg Pres Cons Cons Grp Targ Frc Gas Usage
GR1 GV 1 1* 5000 100 30 GR1 GA 0.5 2* MUGAS PROD /
```

**Keywords**

973
Tables of gas plant recovery factors

This keyword enables tables to be entered to control gas plants. (See "Gas plants and NGLs" in the ECLIPSE Technical Description.)

The keyword contains the following items of data.

1. The gas plant table number.
   This should lie between 1 and the maximum number of gas plant tables, specified using the GPTDIM keyword in the RUNSPEC section.

2. The lower component index used to define the heavy fractions.
   This defaults to the last component.

3. The upper component index used to define the heavy fractions.
   This defaults to the last component.

4. A table of recovery fractions versus heavy mole fraction.
   Each value of the heavy mole fraction should be followed by $N_c$ values (where $N_c$ is the number of components in the run). The set of $N_c$ values lists the liquid recovery fractions for each hydrocarbon component (that is the number of moles of liquid per unit mole of feed for each component).

   Thus there should be $N_c + 1$ numbers for each value of the heavy mole fraction.

   The table should be terminated with a slash.

   The defaults for items 2 and 3 imply that the gas plant table is normally of recovery fractions against the mole fraction of the last component in the feed.

   If the recoveries do not vary with the heavy mole fraction, it is only necessary to enter data for a single value of that heavy mole fraction.

**Example**

The first column contains the heavy component mole fractions, for the tenth component. The remaining columns are the recovery fractions of components 1 to 10.

```
GPTABLE
1 10 10
-- C7+  CO2  N2  C1  C2  C3  C4-6  C71  C72  C73  C74
 0.0  0.0  0.00  0.00  0.00031  0.05  0.10  1.0  1.0  1.0  1.0
 0.2  0.0  0.00  0.00  0.00032  0.07  0.12  1.0  1.0  1.0  1.0
/
```
GPTABLEN  Gas plant table with NGL recoveries

This keyword enables tables to be entered to control gas plants. (See "Gas plants and NGLs" in the ECLIPSE Technical Description.)

The keyword contains the following items of data:

1. The gas plant table number.
   This should lie between 1 and the maximum number of gas plant tables, specified using the GPTDIMS keyword in the RUNSPEC section.

2. The lower component index used to define the 'heavy' fractions.
   This defaults to the last component.

3. The upper component index used to define the 'heavy' fractions.
   This defaults to the last component.

4. A table of recovery fractions versus heavy mole fraction.
   Each value of the heavy mole fraction should be followed by 2 N_c values (where N_c is the number of components in the run). The first set of N_c values lists the oil recovery fractions for each hydrocarbon component (that is the number of moles of oil per unit mole of feed for each component). The second set of N_c values lists the NGL recovery fractions.

   Thus there should be 2 N_c +1 numbers for each value of the heavy mole fraction.

The table should be terminated with a slash.

The defaults for items 2 and 3 imply that the gas plant table is normally of recovery fractions against the mole fraction of the last component in the feed.

If the recoveries do not vary with the heavy mole fraction, it is only necessary to enter data for a single value of that heavy mole fraction.

Example

The first column contains the heavy component mole fractions, for the ninth component. The remaining columns are the recovery fractions of components 1 to 9.

```
GPTABLEN 1 9 9
--Z9 CO2 N2 C1 C2 C3 C4-6 CZ7 CZ8 CZ9
0.0 0.00 0.00 0.00 0.0031 0.05 1.0 1.0 1.0 --Oil recovs at Z9=0
0.01 0.01 0.01 0.05000 0.02 0.01 0.0 0.0 0.0 --NGL recovs at Z9=0
0.2 0.00 0.00 0.00 0.00032 0.05 1.0 1.0 1.0 --Oil recovs at Z9=0.2
0.01 0.01 0.01 0.05000 0.02 0.01 0.0 0.0 0.0 --NGL recovs at Z9=0.2
/```

Keywords
GPTABLE3  **Gas plant table with oil, NGL and gas recoveries**

This keyword enables tables to be entered to control gas plants with recovery factors specified for oil, NGL and gas. (See “Gas plants and NGLs” in the ECLIPSE Technical Description.)

The keyword contains the following items of data.

1. **The gas plant table number**
   
   This should lie between 1 and the maximum number of gas plant tables, specified using the GPTDIMS keyword in the RUNSPEC section.

2. **The lower component index used to define the 'heavy' fractions**
   
   This defaults to the last component.

3. **The upper component index used to define the 'heavy' fractions**
   
   This defaults to the last component.

4. **A table of recovery fractions versus heavy mole fraction.**
   
   Each value of the heavy mole fraction should be followed by 3 $N_c$ values (where $N_c$ is the number of components in the run). The first set of $N_c$ values lists the oil recovery fractions for each hydrocarbon component (that is the number of moles of oil per unit mole of feed for each component). The second set of $N_c$ values lists the NGL recovery fractions and the third set of $N_c$ values lists the gas recovery fractions. The sum of these recovery fractions for each component need not equal unity.

   Thus there should be $3 N_c + 1$ numbers for each value of the heavy mole fraction.

   The table should be terminated with a slash.

   The defaults for items 2 and 3 imply that the gas plant table is normally of recovery fractions against the mole fraction of the last component in the feed.

   If the recoveries do not vary with the heavy mole fraction, it is only necessary to enter data for a single value of that heavy mole fraction.

**Example**

The first column contains the heavy component mole fractions, for the ninth component. The remaining columns are the recovery fractions of components 1 to 9.

```plaintext
GPTABLE3
1 9 9
  --Z9 CO2 N2 C1 C2 C3 C4-6 CZ7 CZ8 CZ9
  0.0  0.00  0.00  0.00031  0.05  0.10  1.0  1.0  1.0  --Oil recovs at Z9=0
  0.95  0.99  0.99  0.94969  0.93  0.89  0.0  0.0  0.0  --Gas recovs at Z9=0
  0.2  0.00  0.00  0.000032  0.05  0.10  1.0  1.0  1.0  --Oil recovs at Z9=0.2
  0.01  0.01  0.01  0.05000  0.02  0.01  0.0  0.0  0.0  --NGL recovs at Z9=0.2
  0.95  0.99  0.99  0.94969  0.93  0.89  0.0  0.0  0.0  --Gas recovs at Z9=0.2
/
```

Keywords

976
Gas plant table dimensions

The data consists of the following items, describing the dimensions of the gas plant and recovery plant table data to be used in the run.

1. Maximum number of gas plant tables.
   This includes both tables specified with GPTABLE, GPTABLEN and GPTABLE3.
   DEFAULT: 1

2. Maximum number of rows of gas plant table data.
   DEFAULT: 1

3. Maximum number of rows of recovery plant table data.
   This specifies the maximum number of rows in item 3 of keyword RECOVERY.
   DEFAULT: 2

See "Gas Plants and NGLs" in the ECLIPSE Technical Description for further information.

Example

Two gas plant tables, each with a maximum of twenty rows.

```
GPTDIMS
2 20 /
```
GQUALITY  Set group gas quality target

In GASWAT mode it is possible to set a target for the quality of gas produced by a group. The quality is in terms of calorific value/mole. Calorific value is determined by summing the component rates multiplied by their calorific values set with the keyword CALVAL, and then dividing by the total molar rate. (See "GASWAT option" in the ECLIPSE Technical Description for further information.)

The group must have a target such as a gas rate, which is distributed between the sons using guide rates. By modifying these guide rates we can weight the production between low and high quality wells so that a required group quality is obtained as well as satisfying the group production target. The quality target is only satisfied if:

1. At least two sons are under group control.
2. At least one son must be producing gas with a higher quality than the GQUALITY value.
3. At least one son must be producing gas with a lower quality than the GQUALITY value.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The set of records is ended by a blank record, containing only a slash (/).

1. Group name or name root, or FIELD (to set the field quality target)
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Gas quality target
   UNITS: kJ/kg-M (METRIC), Btu/lb-M (FIELD), J/gm-M (LAB), kJ/kg-M (PVT-M)

Example

```
GQUALITY
  FIELD  1.51 /
```

Keywords
GRADGRUP  Specify field and group quantities to be output to HMD file

This keyword can be used to specify which field and group quantities are written to the HMD file.

The keyword must be followed by a set of records specifying mnemonics associated with derivatives. Each record should be terminated with a slash (/) and the set of records must end with a slash.

The available mnemonics are:

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>GOPR</td>
<td>Surface group oil rate</td>
</tr>
<tr>
<td>GWPR</td>
<td>Surface group water rate</td>
</tr>
<tr>
<td>GGPR</td>
<td>Surface group gas rate</td>
</tr>
<tr>
<td>GWCT</td>
<td>Group water cut</td>
</tr>
<tr>
<td>GGOR</td>
<td>Group gas-oil ratio</td>
</tr>
<tr>
<td>GOPT</td>
<td>Surface group oil production cumulative total</td>
</tr>
<tr>
<td>GWPT</td>
<td>Surface group water production cumulative total</td>
</tr>
<tr>
<td>GGPT</td>
<td>Surface group gas production cumulative total</td>
</tr>
</tbody>
</table>

The keyword is optional. If it is not specified, no group of field derivatives is produced. If the keyword is used more than once, only mnemonics from the most recent invocation are used so that the list may be specified again at a restart or during the run.

Output from ECLIPSE is for any groups specified using the RPTHMG keyword.

Example

Output derivatives only for group oil and water rates:

```
GRADGRUP
  'GWPR' /
  'GOPR' /
```

Keywords

979
Specify derivatives of solution quantities to be output to the restart file

This keyword can be used to specify which gradients of solution data are output to the restart files. The keyword is entirely optional. If it is not specified then no gradients of solution data are written to the restart file.

Note: The eclipse4d license feature is required to use this keyword. If this feature is not available then no output will be generated. Please contact your SIS account manager to obtain a license.

The keyword must be followed by a set of records specifying mnemonics associated with derivatives. Each record should be terminated with a slash (/) and the set of records must end with a slash.

The available mnemonics are currently:

- PRES  output gradients of grid block oil pressures
- SOIL  output gradients of grid block oil saturation
- SGAS  output gradients of grid block gas saturation
- SWAT  output gradients of grid block water saturation
- DENO  output gradients of grid block oil phase density
- DENG  output gradients of grid block gas phase density
- DENW  output gradients of grid block water phase density
- PORO  output gradients of grid block reservoir porosity
- RS    output gradients of solution GOR
- ACIP  output gradients of grid block pressure-wave impedance
- ACIS  output gradients of grid block shear-wave impedance
- POIS  output gradients of grid block Poisson’s ratio

Additional mnemonics (for example for fluid properties or a combination of fluid and rock properties) could easily be output if requested, for instance effective densities. Please contact SIS with any requests or suggestions.

Note: The gradients output will be encoded unless the UNCODHMD keyword is used. The use of this keyword requires the uncodedhmd license feature to be available. Your Schlumberger account manager will be pleased to help you in obtaining a license.

Example

This example shows how to request the output of gradients of pressure, water saturation and pressure-wave impedance for all gradient parameters in the study.
### Formats

A record ‘GRADRESV’ is written to the HMD and Restart files to list the mnemonics. For example, referring to the above example

```
GRADRESV
            PRES'
            SWAT/
            'ACIP'
            /
```

or

```
GRADRESV
            4 'CHAR'
            'GPRES' 'GSOIL' 'GSWAT' 'GSGAS'/
```

The solution gradient data is written to the Restart file in the following manner (see the *ECLIPSE File Formats Reference Manual* for details of how restart data is written from ECLIPSE).

Header **GRADSOL** indicates a solution gradient mnemonic:

For example

```
GRADSOL
        5 'CHAR'
HMTRANZ 'GRADZONE' '1' '1' '3' /
```

The record **GRADSOL** contains some or all of the following items:

1. **Parameter name**
2. **Keyword name, associated with parameter:**
   - ‘GRADZONE’ for regional parameters defining gradzones
   - ‘FAULTS’ for faults
   - ‘AQUCTS’ / ‘AQUFET’ for aquifers
   - ‘SATNUM’ relative permeability shape parameters
   - ‘PVTNUM’ / ‘SATNUM’ / ‘ROCKNUM’ for rock parameters
   - ‘MULTNUM’ for inter-region transmissibility multipliers
3. **Feature name**
   - Blank for regional parameters
   - Name of fault for fault parameters
   - Blank for all other parameters
4. **Parameter type**
   - This is described under **HMHEAD** in "Output from the Gradient option" in the *ECLIPSE Technical Description*
5. Region ID or Feature number
   This is the subregion number for regional parameters
   Fault number for fault parameters
   Aquifer number for aquifer parameters
   Saturation table number for relative permeability shape parameters
   Rock table number for rock parameters
   First region number for inter-region connection parameters

6. Associated phase
   This is a pointer internal to ECLIPSE and should not be used by an external application.

7. region ID
   This is the second region number for inter-region connection parameters
   Set to zero for all other parameters

8. Not used at present. This is for forward extensibility.

9. Not used at present. This is for forward extensibility.

10. Not used at present. This is for forward extensibility.

Following each ‘GRADSOL’ record, the solution gradients are written in the following form:

```
GRADSOL       6 'CHAR'
HMTRNXY       'GRADZONE'       '3       ' '3       ' '1       '
GPRES_13       300 'REAL'
0.45278375E+03 0.40524387E+03 0.35893173E+03 -0.87420359E+01
-0.12985765E+02 -0.22687635E+02 -0.40819241E+02 -0.70829422E+02
-0.11564088E+03 0.16381274E+03 0.19503807E+03 0.19354639E+03
0.18682153E+03 0.16461343E+01 0.25963242E+01 0.47793212E+01
```

The ‘GPRES_nn’ mnemonic indicates oil pressures. The names in GRADRESV are preceded by the letter ‘G’ so they are distinct from names ‘SGA’, ‘SWAT’ which ECLIPSE and GRAF use for the solution values.

The header is usually of the form ‘G’ //name(1:4)//number; except when the ‘name’ is less than four characters, for example, the solution gas-oil ratio, RS. For this exception, additional underscores are used.

Characters 5 to 8 in the header just make a unique number such that GRAF does not get confused by several headers called ‘GPRES’ for example. The ‘GRADSOL’ header qualifies the actual solution gradients so that the last 3 letters of the header can be ignored. The {number} is required only because applications such as GRAF and FloViz have no special code to recognize these mnemonics.
GRADRFT

Request gradient output of RFT quantities

When using the Gradient option, it is possible to output gradients of RFT quantities to the RFT file for matching measured RFT. Normally, gradient information for well observations is written to the HMD file when the Gradient option is being used. The gradient data will normally be used by the SimOpt program.

The keyword must be followed by a set of records specifying mnemonics associated with derivatives. Each record should be terminated with a slash (/) and the set of records must end with a slash.

The available mnemonics are:

- PRESSURE  Connection grid block pressure
- SWAT      Connection grid block water saturation
- SGAS      Connection grid block gas saturation
- SOIL      Connection grid block oil saturation

The keyword is optional. If it is not specified, no RFT gradients will be output. The list of wells for which RFT gradients are required can be specified in the GRADWELL keyword.

When GRADRFT is specified, RFT output is automatically switched on at report timesteps for wells specified in the RPTHMW keyword, so that it is not necessary to specify the WRFT keyword in addition.

If the keyword is entered more than once, only mnemonics from the most recent entry are used, so that the list may be specified again at a restart or during the run.

Refer to "Gradient option" in the ECLIPSE Technical Description for further information.

Example

Output derivatives only for connection block pressure (oil phase pressure), water saturation and gas saturation.

```
GRADRFT
  PRESSURE /
  SWAT /
  SGAS /
/
```
Specify well quantities to be output to HMD file

This keyword can be used to specify which well quantities are written to the HMD file.

The keyword must be followed by a set of records specifying mnemonics associated with derivatives. Each record should be terminated with a slash (/) and the set of records must end with a slash.

The available mnemonics are:

- **WBP** One-point pressure average - see keyword WPAVE
- **WBP4** Four-point pressure average - see keyword WPAVE
- **WBP5** Five-point pressure average - see keyword WPAVE
- **WBP9** Nine-point pressure average - see keyword WPAVE
- **WOPR** Surface oil rate
- **WWPR** Surface water rate
- **WGPR** Surface gas rate
- **WWCT** Well water cut
- **WGOR** Well gas-oil ratio
- **WBHP** Bottom hole pressure
- **WOPT** Surface oil production cumulative total
- **WWPT** Surface water production cumulative total
- **WGPT** Surface gas production cumulative total

The keyword is optional.

If this keyword is not specified, then the first 9 items listed above (WBP, WBP4, WBP5, WBP9, WOPR, WWPR, WGPR, WWCT, and WGOR) are output for each well.

If this keyword is not specified, then the six items (WOPR, WWPR, WGPR, WWCT, WGOR, and WBHP) are output for each well.

If the keyword is used more than once, only mnemonics from the most recent invocation are used so that the list may be specified again at a restart or during the run.

**Example**

Output derivatives only for water rate, water cut and bottom hole pressure:

```
GRADWELL
  'WWPR' /
  'WWCT' /
  'WBHP' /
```

**Keywords**
GRAVCONS Specify user defined gravity constant

The default gravity constant values are defined in the "Units" in the ECLIPSE Technical Description. If this value needs to be overwritten, it can be done using the GRAVCONS keyword.

The data comprises one value terminated with a slash (/).

UNITs: m^2bars/kg (METRIC), ft^2psi/lb (FIELD), cm^2atm/gm (LAB), m^2atm/kg (PVT-M)

Example

```
GRAVCONS
0.05/
```
**GRAVDR**  

**Use gravity drainage for dual porosity runs**

This requests that gravity drainage and imbibition are modeled between the matrix and fracture cells in a dual porosity run.

If this option is used, then the vertical dimensions for blocks of matrix material should be entered with keywords `DZMTRX` or `DZMTRXV` in the GRID section.

See "Dual porosity" in the *ECLIPSE Technical Description* for further information. See also keyword `GRAVDRM`.

The keyword has no associated data.
**GRAVDRB**  Use the vertical discretized gravity drainage model for dual porosity runs

This keyword requests that the vertical discretized gravity drainage and imbibition model should be used. If this option is used, then the geometry of the matrix sub-cell specified by keyword `NMATOPTS` must be vertical.

See "Dual porosity" in the *ECLIPSE Technical Description* for further information.

The keyword has no associated data.

In ECLIPSE 100, with this model, the minimum value for the number of global cells in the X direction (keyword `DIMENS`) is 2.
Use the alternative gravity drainage model for dual porosity runs

This keyword requests that the alternative gravity drainage and imbibition model should be used. (The GRAVDRM keyword may be used instead of the GRAVDR keyword; if both are present the GRAVDRM keyword supersedes GRAVDR.)

The keyword has one data item, terminated with a slash(/):

1. Allow re-infiltration
   - **YES** Oil flow is allowed both out of and into the matrix block.
   - **NO** Oil flow is only allowed out of the matrix block. With no re-infiltration the final recovery from the matrix is more predictable.

      DEFAULT: **YES**

2. See "Dual porosity" in the *ECLIPSE Technical Description* for further information.
**Fluid gravities at surface conditions**

The data comprises NTPVT (see keyword TABDIMS in the RUNSPEC section) records, each terminated by a slash (/).

Each record consists of the following items of data:

1. Oil API gravity.
   - DEFAULT: 45.5
   - UNITS: °API

2. Water specific gravity (with reference to pure water).
   - DEFAULT: 1.0

3. Gas gravity (with reference to air).
   - DEFAULT: 0.7773

In compositional runs the oil and gas surface densities are obtained from the equation of state and the values entered under the GRAVITY keyword are not used. A dummy value may be entered in this case. The water specific gravity is always required. Note that GRAVITY always contains three items, even if not all three phases are present in the run.

**Note:** When using the API tracking option (keyword API in the RUNSPEC section), the set of records must be entered in decreasing order of oil API gravity.

Keyword DENSITY can be used as an alternative to GRAVITY.

See "Units" of the ECLIPSE Technical Description for conversion formulae between gravity and density.

**Example**

With two PVT tables:

```
GRAVITY
  40.1 1.16 0.852 /
  36.5 1.16 0.852 /
```
GRDREACH  Defines location of grid blocks connecting to a river-bed

This keyword defines the location of the grid blocks connecting to a river and instructs ECLIPSE to allocate each ‘completion’ to a river reach. The river’s reach structure must previously have been defined with the REACHES keyword. The location of each connected grid block is defined within the river by the branch name and the distance down the river to the start and end of the grid-block. This information must be supplied for all the grid-blocks connecting to the river that have been specified with the COMPRIV keyword. ECLIPSE allocates each grid block connection to the reach whose nodal point lies nearest to the center of the grid block. It is also possible to specify which reach a grid block belongs to.

The keyword can only be used on one river at a time, thus it must be specified more than once if there is more than one river.

The keyword is followed by a number of data records to define the location of all the grid block connections in the river. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

The data items in each record are:

**Record 1**

The first record consists of a single item of data that identifies the river.

1. Name of the river

End the data record with a slash (/).

**Each subsequent record**

The subsequent records set the start and end of each grid block in terms of distance along the river from the river’s zero reference point, and the branch number. The zero length reference point is defined relative to item 5 of the first record of keyword REACHES. The grid blocks must previously have been defined with keyword COMPRIV.

The grid blocks may either be specified individually, or in one or more “ranges”. A “range” is a contiguous row or column of grid blocks that are connected to the same branch of the river. The river is assumed to go through each grid block in the specified direction. The distance along the river to the start and end of each grid block in the range is determined by summing the DX, DY or DZ thicknesses of the grid blocks in the range. All grid blocks in the river must have their distances set, either individually or within a range, working outwards along the main stem and each branch towards its source.

1. I - location of the individual grid block, or the grid block at the start of the range (nearest the river mouth)
2. J – location of the individual grid block, or the grid block at the start of the range (nearest the river mouth).
3. K - location of the individual grid block, or the grid block at the start of the range (nearest the mouth). This item will usually be 1.
4. Branch name on which this grid block (or range of grid blocks) is located.
5. Distance down the river from the river’s zero reference point to the start of this grid block.

**UNITS**: m (METRIC), ft (FIELD), cm (LAB)
DEFAULT: Zero, for the first of these records, or the distance to the end of the previous grid block or range, for subsequent records

6. Distance down the river from the river’s zero reference point to the end of this grid block.
   This item, if set, must be greater than item 5.
   UNITS: m (METRIC), ft (FIELD), cm (LAB)
   DEFAULT: Value of item 5 + thickness of the grid block in the direction of penetration

7. Reach number the grid blocks connect to
   Optional: If entered this value will override any value calculated by ECLIPSE. It is a particularly good idea to use this keywords if the grid blocks are dead grid blocks.

8. Direction of penetration through the grid block or the range.
   X or I for horizontal penetration in the x-direction
   Y or J for horizontal penetration in the y-direction
   Z or K for vertical penetration
   This item is used for calculating the distance to the end of the grid block (or the grid blocks in the range), where it defines which component of the grid block size to add (DX, DY or DZ). The item must be set if item 6 is defaulted, or if a range is specified in item 9. Otherwise it may be defaulted as it is not required.

9. Grid block I, J, or K coordinate at the end of the range, depending on the direction of penetration set in item 7. (For example the I-coordinate of the grid block at the end of the range, if the branch penetrates in the I-direction.)
   This may be greater than, or less than, the corresponding grid block coordinate in item 1, 2 or 3, depending on whether the branch penetrates out towards the positive or negative I, J or K direction.
   If this item is set, the lengths are calculated for all the river’s grid blocks that lie within the range. Should the range pass through an inactive grid block ECLIPSE prints a warning message and a thickness must be set.
   If this item is defaulted, this record sets the data for the single grid block identified by items 1, 2 and 3.

10. Contact area of this grid block with the river (See item 17)
    UNITS: m² (METRIC), ft² (FIELD), cm² (LAB)

11. Table number describing contact areas as a function of the depth of water in the river. (See RBEDCONT)
    One of items 10 or 11 must not be defaulted. If both are entered the table number will be ignored.

12. Productivity index of grid block with river (See item 17)
    UNITS: m/day/bars (METRIC), ft/day/psi (FIELD), cm/hr/atm (LAB)
    DEFAULT: 0.0

13. The length of a dead grid block in the penetration direction
    UNITS: m (METRIC), ft (FIELD), cm (LAB)
14. Option to connect a REACH either to the cell center (default) or to cell top. A value of 1 connects this REACH to cell top, and a value of 0 connects this REACH to cell center.
   DEFAULT: 0

15. Adjustment of REACH connection elevation with positive value pointing upward. This item allows a REACH to connect anywhere within a cell by introducing a shift to the connection point.
   DEFAULT: 0

16. Remove capillary pressure from inflow equation? ‘YES’ or ‘NO’
   DEFAULT: ‘NO’

17. Choice of infiltration equation? The choice of infiltration equation should either be zero or one depending on the choice of model. If the default is used you must enter items 10 and 12, otherwise items 18 and 19 must be entered. See “Rivers” in the ECLIPSE Technical Description for more details on the river infiltration equations.
   DEFAULT: 0

18. Hydraulic conductivity (See item 17)
   UNITS: m/day (METRIC), ft/day (FIELD), cm/hr (LAB)
   DEFAULT: 0.0

19. Thickness of river bed (See item 17)
   UNITS: m (METRIC), ft (FIELD), cm (LAB)
   DEFAULT: 0.0

End the data record with a slash (/).

End the set of data records with a slash.

Note: River data can be complicated to enter. It is advisable to check your data by examining the output generated in the Print file.
The output is generated by setting RPTSCHED mnemonic WELSPECS.

Example

```
GRDREACH
  MYRIVER /
 35 1 1 MYRIVER 2* 1 2* 100 0 0.01 200 2*/
 35 2 1 MYRIVER 2* 2 2* 100 0 0.01 200 2*/
 35 3 1 MYRIVER 2* 3 2* 100 0 0.01 200 2*/
 35 4 1 MYRIVER 2* 4 2* 100 0 0.01 200 2*/
 35 5 1 MYRIVER 2* 5 2* 100 0 0.01 200 2*/
 35 6 1 MYRIVER 1000 1* 1* 'J' 8 100 0 0.01 200 2*/
 35 9 1 MYTRIB 2* 1* 'J' 13 120 0 0.02 1 -0.5/
 35 9 1 MYTRIB 2* 1* 'J' 13 120 0 0.02 1 -0.5/
```

Keywords

992
Reference gravities for EoS

In a run with \( N_c \) components this keyword specifies the fluid gravities at the reference conditions with respect to that of water. The keyword can be used either in a run which uses the Zudkevitch Joffe equation or in a thermal run (keyword `THERMAL` in the `RUNSPEC` section). Alternatively the keyword `DREF` can be used.

The density of water is:

- **UNITS**: 999.014 kg/m\(^3\) (METRIC), 62.3664 lb/ft\(^3\) (FIELD), 0.999014 gm/cc (LAB), 999.014 kg/m\(^3\) (PVT-M)

There should be one value for each component.

**DEFAULT**: 1.0

If multiple equations of state are being used, a record should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 `TABDIMS`. The use of multiple EoS regions is discussed in "Equations of state regions" in the *ECLIPSE Technical Description*.

**Example**

```
GREF
0.805 0.778 0.454 0.548 0.610 0.659 0.719 0.838 0.907 /
```
GREFS

Reference gravities for surface EoS

This optional keyword supplies component reference gravities for use in the surface calculations. The default, if this is not supplied, is for these to follow the reservoir values supplied with GREF.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.
GRIDFILE

Control output of the grid geometry file

The keyword should be followed by up to two integers.

This value controls the amount of data written to the GRID geometry file. The possible values are:

1. Control of the GRID file output
   - 0 No GRID file is produced
   - 1 A standard GRID file is produced. (for active cells only).
   - 2 An extended GRID file is produced, containing:
     - Inactive cell data
     - Non-neighbor connection data
     - Local grid refinement data
     - Local grid coarsening data
   
   DEFAULT: 0

   If any local grid refinements or coarsening are present, then a value of 1 will be promoted to 2.

2. Control of the extensible grid file output (EGRID)
   - 0 No EGRID file is produced
   - 1 Output an extensible grid file, containing:
     - Inactive cell data
     - Non-neighbor connection data
     - Local grid refinement data
     - Local grid coarsening data
   
   DEFAULT: 1

The data must be terminated by a slash (/).

If neither the GRIDFILE nor NOGGF keywords are used, then by default, an EGRID file will be produced.

If the NOGGF keyword is used then no GRID file of either form is output.

The Extensible Grid file (EGRID file) contains the same information as the standard grid file in corner point or block centered grids. The format is typically more efficient for generating a smaller file that is faster to load into post-processing programs. In addition the format is used to store unstructured grid data, where ECLIPSE does not know original grid geometry but just the processed pore volumes and transmissibilities.

The GRAF program can read the extensible EGRID file, the standard GRID file or the extended GRID file. However, an EGRID file or an extended GRID file is required if the grid displays are to include the local grid refinements.
Note: Setting the first item in GRIDFILE to 2 is recommended for correct display in 3D graphical applications.

### Examples

#### Example 1

```
-- Extensible EGRID file only
GRIDFILE
0 1 /
```

#### Example 2

```
-- Both EGRID and Extended GRID file
GRIDFILE
2 1 /
```
## GRIDOPTS

### Options to process grid data

This keyword requests additional options for processing the grid data. It should be followed by some or all of the following items, terminated with a slash (/).

1. Allow the alternative transmissibility multipliers including `MULTX-`, `MULTY-`, `MULTZ-` (which apply to the grid block faces X-, Y-, Z-) to be used in the GRID, EDIT or SCHEDULE sections. Similarly, the keyword is required if the alternative diffusivity multipliers, which include `DIFFMX-`, `DIFFMY-`, `DIFFMZ-`, are used.

   **YES** Keywords `MULTX-`, `DIFFX-` and related keywords may be used

   **NO** Otherwise

   **DEFAULT:** NO

2. `NRMULT` The maximum number of `MULTNUM` regions entered in the GRID section, which apply either to inter-region transmissibility multipliers using the `MULTREGT` keyword, or pore volume multipliers using the `MULTREGP` keyword. For both cases, if this item is set to zero or defaulted then any multiplier is applied between flux regions entered using `FLUXNUM`.

   **ECLIPSE 100**

   `MULTNUM` regions can similarly be used for diffusivity and thermal conductivity multipliers (`MULTREGD` and `MULTREGH`).

   **DEFAULT:** 0

   **ECLIPSE 100 only**

   **Note:** If this value is defaulted or set to 0 and the 4th item of `REGDIMS` is also defaulted or set to 0, the simulator behaves as though all grid blocks are in `MULTNUM` region 1. This has no effect except that specifying multipliers for region 1 will not generate a warning or error, and is a side-effect of the default value for `MULTNUM` being 1.

3. `NRPINC` The maximum number of `PINCHNUM` regions entered in the GRID section, which apply to the calculation of pinchouts by region.

   **DEFAULT:** 0

   **ECLIPSE 100 only**

   **Note:** Data item 3 is not available for use in this release. Any values read in for this item is ignored by the simulator.

   **ECLIPSE 300**

   **Note:** This keyword can be specified once only, unless item 223 in OPTIONS is set to 1 for backward compatibility purposes. Users who do this should be aware that while respecifying it will overwrite the values, it does not undo options which were triggered by the first specification, in particular the choice of which region set will be used by default for `MULTREGT`.

### Keywords

- `MULTX-`
- `MULTY-`
- `MULTZ-`
- `DIFFMX-`
- `DIFFMY-`
- `DIFFMZ-`
- `MULTREGT`
- `MULTREGP`
- `MULTREGD`
- `MULTREGH`
Example

| GRIDOPTS | YES 0 3 / |

Keywords

998
GRIDUNIT

Specifies the grid data units

This keyword is entirely optional; it is normally output by grid pre-processors.

The keyword should be followed by up to two items of data:

1. Unit of length for the grid data
2. A flag set to MAP or left blank

   This is to indicate whether the grid data are measured relative to the map, or relative to the origin given by the MAPAXES keyword.

The data must be terminated by a slash (/).

This data is then available through the GRID file for post-processing.

Example

```
GRIDUNIT
    METRES MAP /
```

Note: ECLIPSE 100 checks if the units specified in the INCLUDE or IMPORT file are consistent with the units specified in the RUNSPEC section of the dataset. If necessary, it then does the appropriate conversion assuming an origin at (0,0,0). The conversion is only applied up to the end of the next INCLUDE / IMPORT file (so care needs to be taken if the data grid is split into separate files), or until the end of the GRID section. If this is not satisfactory, check that the GRIDUNIT keyword is correct. If necessary, export the data again from the gridding software using the correct units.

Note: ECLIPSE 300 checks if the units specified in the INCLUDE or IMPORT file are consistent with the units specified in the RUNSPEC section of the dataset. Unlike ECLIPSE 100, it does not convert the geometric grid and edit data. Export the data from the gridding software again using the correct units, and check that the GRIDUNIT keyword is correct.
GRUPFUEL  Specifies group fuel rate

This keyword specifies the amount of gas produced from a group that is burnt for fuel. This reduces the gas available for reinjection.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/). The set of records is ended by a blank record, containing only a slash.

1. Group name or name root
   or FIELD (to set the field fuel rate)
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. The constant usage rate. This is summed with the quantity specified using item 3 to give the overall gas fuel requirement. A user defined argument can be set with keywords UDQ and UDADIMS.
   UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT–M)
   DEFAULT: 0.0

3. The fractional gas rate. This is a fraction of the total gas production from the group (before any fuel, sales and re-injection requirements have been taken). This fractional part is calculated at each timestep and then summed with the quantity specified using item 2 to give the overall gas fuel requirement. A user defined argument can be set with keywords UDQ and UDADIMS.
   DEFAULT: 0.0

4. The component template.
   This may be used to alter the composition of the gas removed for fuel.
   One value is required for each component, the default being 1. If a value less than one is specified, the amount of that component used for fuel will be correspondingly reduced.

   The method used to determine the amount of fuel burnt is:
   • Sum the fuel requirements from any subgroups.
   • If a fractional usage rate is defined (item 3), add the specified fraction of the remaining gas from the group, multiplying the volume rate of each component by its template value. Note that the template can alter the overall volume rate of gas used for fuel when a fractional usage rate is specified.
   • If a constant usage rate is defined (item 2), add this to the fuel usage, subject to an upper limit equal to the remaining gas rate. The component template is used to multiply the contribution from each component. Note that the component template does not alter the constant usage rate, as the component volume rates are normalized again to obey the specified constant rate.
   • The component rates used for fuel are now subtracted from the gas available for subsequent requests such as sales or reinjection. Note that using a component template can alter the composition of the remaining available gas.

   In a multilevel group hierarchy the remaining available gas is reduced for all children of the group concerned. The same reduction fraction is used for all children as for the group concerned - the gas used for fuel is assumed to be evenly taken from the gas produced by the group.

   By default, the group fuel requirement is satisfied prior to the sales requirement. This ordering may be changed using the WTAKEGAS keyword.
The fuel requirements are satisfied from the bottom of the group hierarchy to the top. See also "Group injection control options" in the ECLIPSE Technical Description.

**Note:** For accurate group gas accounting, there is a limitation that reinjection control and any sales gas or fuel gas requirements must all be specified for the same group, usually the FIELD group, and not split across the group hierarchy. This limitation only applies if the injection gas contains group vapor, with the vapor taken from a separator stage (that is not from the whole separator).

**Example**

Set field gas fuel rate of 1000 Mscf/day:

```
GRUPFUEL
FIELD 1000 /
/```

**Keywords**

1001
Specifies group guide rates

This keyword specifies the user guide rates for a group.

The keyword is retained for back-compatibility; you are encouraged to use keywords `GCONPROD` and `GCONINJE` to set group controls and guide rates, which are compatible with both simulators.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Group name or name root
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. A character specifying if group is available for group control.
   This must be start with Y or N.
   DEFAULT: Y

3. The guide rate value.

4. The type of guide rate being specified.
   This must be one of the following:

   - OIL: Oil production guide rate
   - WAT: Water production guide rate
   - GAS: Gas production guide rate
   - LIQ: Liquid production guide rate
   - COM: Linearly combined production guide rate (coefficients must be set with `LINCOM`)
   - WG: Wet gas production guide rate
   - VP: Voidage production guide rate
   - IG: Gas surface volume injection guide rate
   - IW: Water surface volume injection guide rate
   - VG: Gas voidage injection guide rate
   - VW: Water voidage injection guide rate
   - VI: Injection voidage guide rate (sets VG and VW)
   - RV: Reservoir voidage guide rate (sets VP, VG and VW)

Keywords

1002
INJ  Injection guide rate (sets IG and IW)

NONE  No guide rate specified

POTN  A group's guide rates will be set at the beginning of each timestep equal to the group's production and injection potentials - that is the sum of the production and injection potentials of all its subordinate open producers and injectors

ALL  Set all guide rates

DEFAULT: NONE

In general, it is best to either specify all the guide rates for the children of a group (to specify the required distribution of targets explicitly), or to use POTN to instruct potential rates to be used as guide rates.

It is usual to set one production guide rate (one of OI, WA, GA, LI, WG, VP), one water injection guide rate (one of IG, VG) and one gas injection guide rate (one of IW, VW). The others in each of these three classes are set using flow ratios from the value specified. The VI, RV, INJ and ALL options allow several guide to be set explicitly.

See also "Group production control options" in the ECLIPSE Technical Description for further information.

**Example**

```plaintext
GRUPGR
  Z45 Y 100.0 INJ /
```
GRUPINJE  Specifies group injection targets

This keyword specifies the injection control data for a group. Both the nature of the fluid to be injected, and the target and limiting rates at which injection is to occur are defined.

The keyword is retained for back-compatibility; you are encouraged to use keyword GCONINJE instead, which is compatible with both simulators.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Group name or name root
   or FIELD (to set field injection controls)

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. A character string specifying the fluid to be injected.

   In black oil mode this may only be WAT or GAS. In compositional mode, the nature of the injected fluid (that is the mole fraction of each component) must be known. This may be defined in one of the following ways:

   • the injection stream has a user specified composition, defined with the WELLSTRE, WINJMIX or WINJORD keyword

   • the injection stream is taken from the production of a named well or group.

The fluid string must be one of the following:

WAT  Water.

GAS  Gas may be selected in either a black oil or a compositional run.

   In a compositional run the composition of the injected gas is set to that of the field separator gas. However, it is generally better to specify the injected gas composition more precisely.

   From 2007.1, in a compositional run, GAS is internally converted to GV (for the field separator) which has better reinjection and gas accounting behavior. Item 113 in OPTIONS3 can be used to recover the pre-2007.1 behavior.

ST  Implies that the molar composition of the injected fluid has been defined using the WELLSTRE keyword.

MIX  Implies that the molar composition of the injected fluid has been defined as a mixture using either the WINJMIX or WINJORD keywords.

WV  Implies that the composition of the injected fluid is to be taken from the vapor production of a nominated well.
GV  Implies that the composition of the injected fluid is to be taken from the vapor production of a
nominated group.

Only the first two characters are significant.
The injection fluid string may only be defaulted if the group is under group control, in which case the
injection fluid will be supplied by the parent group.

3. A character string is required if the composition of the injection fluid was specified using the ST, MIX,
WV or GV methods.
In these cases this item is a character string specifying the name of the wellstream, mixture, well or
group that defines or supplies the injection stream.

4. A character string specifying the method used to control the rate at which fluid is injected.
This must be one of the following:

NONE  No immediate control of injection rate
WA    Inject a target water volume rate set in item 6
GA    Inject a target gas volume rate set in item 7
RV    Inject the fluid to make up a target reservoir volume injection rate set in item 8
WG    Inject the fluid at a target wet gas rate set in item 9
RE    Reinjection or voidage replacement control, at a fraction set in item 10
GR    Group is immediately under control from a higher level group, injecting its share of the higher
group’s target rate, according to its guide rate (which must be set in the GRUPGR keyword)

The character string determines the rate to be used as an initial target for group injection; other rates
entered are interpreted as limits.

5. The target/limit oil volume rate.
UNITs: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT−M)

6. The target/limit water volume rate.
UNITs: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT−M)

7. The target/limit gas volume rate.
UNITs: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT−M)

8. The target/limit reservoir volume rate.
UNITs: rm$^3$/day (METRIC), rb/day (FIELD), rcc/hr (LAB), rm$^3$/day (PVT−M)

9. The target/limit wet gas injection rate.
Wet gas rate is related to the moles injected using a unity Z-factor at standard conditions.
UNITs: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT−M)

10. The target/limit reinjection fraction or voidage replacement fraction.
If item 15 is set to $S$, item 10 is interpreted as a reinjection fraction. When this is a target, the group attempts to inject the fluid specified in item 2 at a surface rate equal to the reinjection fraction multiplied by the water or available gas rate of the group or well named in item 12. The available gas rate of a group for reinjection is by default equal to the group’s gas production rate plus the rate of any advanced gas associated with the group (keyword GADVANCE) minus its fuel rate (keyword GRUFFUEL) minus its sales target (keyword GRUPSALE); however, the ordering of this definition can be altered using keyword WTAKEGAS. In the case of a reinjection fluid defined by a stream or mixture, the specified fraction of the available surface gas production rate is applied as a wet gas reinjection rate target.

If item 15 is set to $R$, item 10 is interpreted as a voidage replacement fraction. When this is a target, the group attempts to inject the fluid specified in item 2 to make up a total reservoir volume injection rate equal to the voidage replacement fraction multiplied by the total reservoir volume production rate of the group or well named in item 12.

A make-up gas stream may be specified in item 14, which is mixed with the injection stream if the gas injection rate exceeds the available gas rate.

DEFAULT: 1.0 if item 4 is RE, 0.0 otherwise

11. The type of production used to define the rate under reinjection or voidage replacement control.
   This is required for reinjection or voidage replacement control. It may be one of the following:

   $W$ Implies the reinjection or voidage replacement depends on the production of the well named in item 12.

   $G$ Implies the reinjection or voidage replacement depends on the production of the group named in item 12.

   DEFAULT: $W$ if item 2 is $W\!V$, $G$ if item 2 is $G\!V$

12. The name of the production well or group that controls the reinjection or voidage replacement rate.
   This is required for reinjection or voidage replacement control. If item 11 is $W$ it should be the name of a production well. If item 11 is $G$ it should be the name of a group.

   If this name is not specified, it defaults to the fluid composition source name (item 3) for injection fluids $W\!V$ or $G\!V$ in item 2, and to the name of the group in item 1 for the other cases ($W\!A$, $G\!A$, $S\!T$ or $M\!I\!X$ in item 2).

13. Not used at present.

14. The mnemonic for a wellstream to be used as make-up gas.
   This is only required if make-up gas is to be used when the injected gas rate exceeds the advance and/or available produced gas from the source group or well.

15. The way in which reinjection or voidage replacement fraction in item 10 is to be interpreted.

   $S$ Reinject a fraction of surface volume production (that is reinjection)

   $R$ Replace a fraction of reservoir volume production (that is the voidage replacement)

   DEFAULT: $S$

16. The stage of the separator that defines the fluid composition for injection.
The vapor from any stage of the separator may act as a source of injection fluid. The default value of 0 results in the use of the vapor from the whole separator as the injection fluid.

Note that the choice of separator stage also affects the available produced gas supply rate from the source group or well (item 3) if the source is GV or WV (item 2), and also that of the control group or well (item 12) for reinjection.

DEFAULT: 0

Notes

1. To reinject one phase (for example gas) and top it up with another phase (for example water) to achieve voidage replacement, enter two records of GRUPINJE data. The first record sets a target injection rate or reinjection fraction on the first phase. The second record sets a target voidage replacement fraction (usually 1.0) on the second phase. But the target voidage replacement fraction is taken to apply to the total reservoir volume injection of both phases, that is the injection rate of the second phase is controlled to top up the total reservoir volume injection to achieve the voidage replacement target. The second phase is treated as the top-up phase. The top-up phase is taken to be the phase with a target voidage replacement fraction (item 10) or reservoir volume rate (item 8). ECLIPSE 300 will only allow one top-up phase amongst all groups. This is illustrated in Examples 4 and 5.

2. If the injection stream originates from a particular group or well, and the injection target exceeds the available produced gas, the extra gas required is taken from the make-up gas stream, if this is specified in item 14. If no make-up gas is specified, the behavior of the injection controls depends on whether the keyword WAVAILIM is entered. Without this keyword, the injection target is honored by injecting more gas than is available, resulting in a negative excess rate in the gas accounting table. If the WAVAILIM keyword is entered, the injection rate is limited by the amount of gas available for injection from the source group or well. See keyword WAVAILIM for further details.

3. For gas injection wells under group control, it is best to default the fluid injection type defined in items 2 and 3 of their WELLINJE keywords (as illustrated in example 5). This ensures that the injection stream will be that defined by the group. If a gas injection well under group control has its own fluid set, then this fluid, and not that defined by the group, is injected at a rate determined by the group.

4. For water injection wells under group control, the injection fluid WATER must be set in item 2 of their WELLINJE keywords (as illustrated in example 5). If this is not done, then the well defaults to a gas injector and will not contribute to an overall group controlled water injection rate.

5. The reinjection fraction limit (item 10, if item 15 is set to S) ignores makeup gas (item 14). It should not be applied if makeup gas is available, which is treated as an infinite resource.

Examples

Example 1

Replace the voidage produced by the group GRP1 with injected water. The defaulted voidage replacement fraction will automatically be set to 1.0 in this case.

```
GRUPINJE
  GRP1 WATER ' ' RE 10* R /
```

Example 2

Water is to be injected by the group at a rate of 1000 stb/day.
Example 3

The hydrocarbon defined by wellstream MixB is to be injected so as to replace the voidage produced by group PGC.

Example 4

Gas is reinjected from field production at a rate of half the surface production rate. Water tops up the gas to achieve 100% voidage replacement.

Example 5

Gas is reinjected from field production at a fixed rate. Water tops up the gas to achieve a fixed reservoir voidage target.

The WELLINJE keyword defines water and gas injectors, with group controlled rates.
GRUPLIM    Specifies group economic limits

GRUPLIM is an alias for GECON.
GRUPMAST  Identifies the master groups and their slaves

This keyword must be used in a reservoir coupling master run to identify the master groups and to indicate which slaves they are associated with (see "Reservoir coupling" in the ECLIPSE Technical Description).

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. The master group’s name or group name root
   A group name root, ending with an asterisk (*), can be used to refer to several groups in one record.
   The master groups must previously have been declared with the keyword GRUPTREE. They must not contain any wells or have subordinate groups.

2. The name of the slave reservoir containing the group’s associated slave group
   The slave reservoir must previously have been declared and activated with keyword SLAVES. The slave reservoir name is entered as item 1 in that keyword.
   Alternatively, if a default (1*) or blank name (’’) is entered, the group’s flow rates will be read from the input reservoir coupling file instead of being calculated from the slave reservoir simulation. In this case, the USECUPL keyword must be entered to identify the input Reservoir Coupling file, and the group’s slave reservoir should not be included in the SLAVES keyword.

3. The name of the master group’s associated slave group in the slave reservoir
   The associated slave group may have a different name from its master group. For example, the slave group may be the top level group in its slave reservoir, and thus must be called FIELD.
   If a default (1*) or blank name (’’) is entered, ECLIPSE assumes the associated slave group has the same name as its master group.

4. The limiting fraction by which the group’s flow rate may change in a single timestep
   DEFAULT: infinity
   A value may optionally be entered here to limit the fraction by which the group’s flow rate (that is the reservoir fluid volume production rate and its phase injection rates) can change over a single timestep. ECLIPSE does this by restricting the length of the master run’s timestep, based on the previous timestep’s flow rate changes. The restriction takes place retrospectively, and the timestep is not chopped if a sudden rate change causes the limit to be exceeded.
   If the slave groups’ flow rates change significantly over a single timestep of the master run, the group flow rate targets imposed in the master reservoir may not be accurately obeyed. This is because the flow rate targets are apportioned among the master groups according to their slave groups’ flow capabilities at the start of each timestep, while the reported flows are those occurring at the end of the timestep. Reducing the limiting flow change fraction increases the accuracy of the group controls. However, it is also advisable to set a minimum timestep length that this restriction can impose (see keyword RCMASTS), as the fractional flow changes can be large when the flows are small and wells are being opened or closed in the slave reservoirs.
   A value entered here has no effect if the group’s flows are to be read from the Reservoir Coupling file (that is item 2 is defaulted).

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.
### Example

```
GRUPMAST
-- master slave associated limiting
-- group reservoir slave group rate change
-- name name group fraction
LOWER-A LOWER PL-A 0.1 /
LOWER-B LOWER PL-B 0.1 /
UPPER-A UPPER PL-A /
UPPER-B UPPER PL-B /
```

---

**Keywords**

1011
GRUPNET  Defines the standard production network structure

The keyword defines the structure of a standard production network, when using the network option (see "Network option" in the ECLIPSE Technical Description). If you are using the extended network model (keyword NETWORK in the RUNSPEC section), you should define the production network with keywords BRANPROP and NODEPROP instead.

The GRUPNET keyword is followed by any number of records, each containing the following items of data and terminated with a slash (/). The set of records must end with a blank record, containing only a slash.

1. Group name or group name root, or FIELD (to set a fixed pressure for the field node)

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Fixed pressure for the group corresponding to the terminal node of the network

   Default this item, or set it negative, if the group is not a fixed-pressure terminal node in the network.

   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

3. Production VFP table number for the pipeline from the group to its parent group

   A value of 9999 can be entered to indicate that there is no pressure loss in the network branch between the group and its parent group.

   If the group is a fixed-pressure node (that is if a positive value is set for item 2) this item must be defaulted, since a fixed-pressure group is a terminal node and cannot have a pipeline connection to a higher group.

   If this item is defaulted for a group that is not a fixed-pressure node, the group is not part of the network.

   Two or more network pipelines may share the same VFP table if they have similar pressure loss characteristics.

   DEFAULT: 0 Impling the group has no pipeline connection to its parent group.

4. Artificial Lift Quantity (ALQ) used in the pressure loss calculations for the group's pipeline.

   This quantity is used as a look-up parameter for the VFP table in item 3. It may be regarded as the pump or compressor power, according to the definition used in the construction of the table.

   UNITS: These depend on what the ALQ was meant to represent when the table was created. Refer to section "Record 6:ALQ values" in the description of keyword VFPPROD.

   DEFAULT: 0

5. Flag indicating whether the group is a sub-sea completion manifold, so that a group production target will be met by adjusting its wells' THP limits instead of by the standard method of flowing the wells in proportion to their guide rates.

   YES  A group production target is met by adjusting its wells' THP limits, so that all wells may operate at the same THP
A group production target is met by the standard method of group control, and the wells may operate with different THP values.

Any manifold groups that are subordinate to higher level groups having production rate targets (or limits that become rate targets if violated) should be given guide rates in keyword `GCONPROD`, otherwise their wells will not operate at the same THP.

**ECLIPSE 100**

Only well-groups can be manifolds, not higher level groups that are parents to other groups.

The convergence tolerance and iteration limit for the calculation of the manifold group's THP are set in keyword `NETBALAN`. The THP value input in item 10 of keyword `WCONPROD` for the first well in the group is used as the initial guess for the manifold group’s THP in the first timestep. A suitable value entered for this quantity speeds the convergence of the first calculation.

**ECLIPSE 300**

The treatment in ECLIPSE 300 differs from that in ECLIPSE 100. Entering `YES` indicates that the group's outlet branch acts as an automatic choke, and that any production rate target applied to the group is met by adjusting the pressure loss across the choke. The group’s outlet pipeline must be given a VFP table number of 9999 in item 3, so that its only pressure drop is that of the choke. Groups at any level in the tree may be selected, except for the terminal node which has no outlet pipeline.

DEFAULT: NO

6. Flag indicating whether gas-lift gas from the subordinate wells flows through this group’s pipeline.

NO No liftgas flows along the pipeline, only formation gas.

FLO Add the lift gas flows of the subordinate producers to the gas flow rate along the pipeline. The lift gas flow is assumed to be equal to the sum of the ALQ values of the subordinate producers, multiplied by their efficiency factors.

ALQ Set the ALQ value for this pipeline equal to the sum of the ALQ values of the subordinate producers, multiplied by their efficiency factors. The entry in item 4 is ignored.

If the FLO or ALQ option is selected, the Artificial Lift Quantity for the wells (WCONPROD item 12) and the pipeline must be defined as the rate of lift gas injection. This definition must be selected in the VFP i program if it is used to construct the VFP tables.

**ECLIPSE 300**

If FLO is entered for a well group in ECLIPSE 300, the lift gas flows of its wells are included in the network source term and are included in the network flows up to the terminal node irrespective of the item 6 setting for superior groups. (In ECLIPSE 100 FLO must be entered for all the groups through which the lift gas is to flow.)

DEFAULT: NO

**ECLIPSE 100 only**

7. Equivalence the pipeline's ALQ to the surface density?

The pipeline’s ALQ may optionally be made equivalent to the average surface density of either the oil phase or the gas phase flowing along it. ECLIPSE then automatically sets it equal to this value when it looks up the VFP table. The VFP table must have been prepared using the same definition of the ALQ (see keyword `VFPPROD`), and should span the expected range of surface density values. This option is useful when a mixture of oil or gas with different surface densities flows into the network. Obviously the ALQ cannot then also be used to represent a pump or compressor, or to represent the lift gas flow rate in item 6 above, and the pipeline cannot contain an automatic pump/compressor.
DENO  Set the ALQ equal to the average surface density of the oil flowing along the pipeline.

DENG  Set the ALQ equal to the average surface density of the gas flowing along the pipeline.

NONE  Do not equate the ALQ to the oil or gas density. Leave it at the value entered in item 4 (or allow it to be used to represent lift gas or an automatic pump/compressor).

DEFAULT: NONE

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/). End the data record with a slash (/).

Example

```
GRUPNET
  FIELD  200 / 
  PLAT-A  1* 1 / 
  PLAT-B  1* 2 100 / 
   'GR-A*'  1* 3 1* YES / 
  GR-B1  1* 6 1* YES FLO / 
  GR-B2  1* 7 1* YES FLO / 
/ 
```

Keywords

1014
**GRUPPROD**  **Specifies group production target/limits**

This keyword specifies the production targets and limits for a group.

The keyword is retained for back-compatibility; you are encouraged to use keyword `GCONPROD` instead, which is compatible with both simulators.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. **Group name or group name root**
   
   or **FIELD** (to specify field production controls)
   
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. A character string specifying the method used to control the rate at which fluid is produced.

   This must be one of the following:

   **NONE**  No immediate control of production rate
   
   **OIL**   Oil volume rate
   
   **WAT**   Water volume rate
   
   **GAS**   Gas volume rate
   
   **LIQ**   Liquid (oil + water) volume rate
   
   **COM**   Linearly combined volume rate (coefficients must be set with **LINCOM**)
   
   **RV**    Reservoir volume rate
   
   **WGV**   Wet gas volume rate
   
   **GR**    Group is immediately under control from a higher level group, producing its share of the higher group’s target rate, according to its guide rate (which must be set in the **GRUPGR** keyword)

3. The target/limit oil volume rate.

   **UNITS:** sm$^3$/day (**METRIC**), stb/day (**FIELD**), scc/hr (**LAB**), sm$^3$/day (**PVT-M**)

4. The target/limit water volume rate.

   **UNITS:** sm$^3$/day (**METRIC**), stb/day (**FIELD**), scc/hr (**LAB**), sm$^3$/day (**PVT-M**)

5. The target/limit gas volume rate.

   **UNITS:** sm$^3$/day (**METRIC**), Mscf/day (**FIELD**), scc/hr (**LAB**), sm$^3$/day (**PVT-M**)

**Keywords**

ECLIPSE Reference Manual
Note: If, in compositional mode, a group is controlled by gas volume rate, surface volumes are obtained by \[ \frac{Q_g^m R T_{\text{ref}}}{P_{\text{ref}}} \], where \( Q_g^m \) is the molar gas rate, \( R \) the gas constant, \( P_{\text{ref}} \) the standard pressure and \( T_{\text{ref}} \) the standard temperature.

6. The target/limit liquid (oil + water) volume rate.
   **UNITS:** \( \text{sm}^3 \text{/day (METRIC)}, \text{stb/day (FIELD)}, \text{scc/hr (LAB)}, \text{sm}^3 \text{/day (PVT-M)} \)

7. The reservoir volume rate.
   **UNITS:** \( \text{rm}^3 \text{/day (METRIC)}, \text{rb/day (FIELD)}, \text{rcc/hr (LAB)}, \text{rm}^3 \text{/day (PVT-M)} \)

8. The target/limit wet gas rate. (Compositional Only)
   **UNITS:** \( \text{sm}^3 \text{/day (METRIC)}, \text{Mscf/day (FIELD)}, \text{scc/hr (LAB)}, \text{sm}^3 \text{/day (PVT-M)} \)

9. The target/limit linearly combined rate
   **UNITS:** determined by keyword \text{LCUNIT}.

Note: The rate to be used as an initial target for group production is selected by the mode string (item 2); other rates entered are interpreted as limits. If these are exceeded the group changes control mode, the limit becoming a new target and the target a new limit.

See also "Group production control options" in the \textit{ECLIPSE Technical Description} for further information.

**Example**

```plaintext
GRUPPROD
GRPB RV 4* 1000 /
```

**Keywords**
GRUPRIG

Assigns workover and drilling rigs to groups

This keyword is used to assign workover rigs and drilling rigs to individual groups. Each group may have up to five workover rigs and five drilling rigs allocated to it. Several groups may share the same rig, if required. A rig allocated to a high-level group (or the Field) will be available for use by all the wells in its subordinate groups.

A workover rig is necessary to perform automatic workovers if the time taken to perform workovers has been set to a value greater than zero in keyword WORKLIM. The availability of workover rigs will then determine the rate at which automatic workovers can be performed. For each automatic workover on a well, a workover rig is assigned and is made unavailable for other workovers for the duration entered in keyword WORKLIM. A workover on a particular well is postponed if all the workover rigs that can be assigned to it are already occupied until the end of the timestep.

- In ECLIPSE 100, a workover refers to the automatic closure, plug-back or opening of a well connection resulting from instructions specified in keywords WECON, CECON, GECON, GCONPROD, GCONPRI, GCONSALE, PRORDER and WORKTHP.
- In ECLIPSE 300, a workover refers only to the automatic closure of a well connection as a result of an economic or rate limit being broken.

Connection changes specified manually (for example with COMPDAT) are not affected by rig availability, and do not occupy any rigs. If item 3 of keyword WDRILTIM is set to YES for a given well, the well is treated as closed for the duration of the workover. This is done by temporarily reducing the well's efficiency factor, so that its flows are not included in the group totals and cumulative totals while it is being worked over. If the WORKLIM keyword is absent, workovers take up no rig time, and workover rig availability is not a restriction.

A drilling rig is necessary to open a new well automatically if the time taken to drill the well has been set to a value greater than zero in item 2 of keyword WDRILTIM. The availability of drilling rigs will then determine the rate at which new wells can be opened automatically. When a new well is to be drilled, a drilling rig is assigned and is made unavailable for drilling other wells for the duration entered in keyword WDRILTIM. The drilling of a well is postponed if all the drilling rigs that can be assigned to it are already occupied until the end of the timestep. The drilling processes controlled by rig availability are:

- Wells opened from the Drilling Queue (see keywords QDRILL and WDRILPRI)
- Wells on Automatic Opening (for example with item 2 in keyword WCONPROD set to AUTO)
- Wells that are opened on the closure of another well (see item 9 of keyword WECON).

Wells opened manually are not affected by rig availability, and do not occupy any rigs. If item 3 of keyword WDRILTIM is set to YES for a given well, the well is treated as closed for the duration of the drilling process. This is done by temporarily reducing the well's efficiency factor, so that its flows are not included in the group totals and cumulative totals while it is being drilled. If the WDRILTIM keyword is absent, drilling takes up no rig time, and drilling rig availability is not a restriction.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root
   or FIELD (if the rig is being assigned to the whole field)
   A group name root, ending with an asterisk (*), can be used to refer to several groups in one record.
2. Workover rig number (between -2 and 99)
   1-99   A workover rig with this number is added to (or removed from) the group
   0     No change to existing workover rigs
   -1    A workover rig with this number prevents workovers on any well subordinate to the group,
       regardless of the availability of other rigs
   -2    This number causes the removal of all workover rigs currently belonging to the group (but not
       any belonging to subordinate or superior groups).

   DEFAULT: 0

3. Drilling rig number (between -2 and 99)
   1-99   A drilling rig with this number is added to (or removed from) the group
   0     No change to existing drilling rigs
   -1    A drilling rig with this number prevents any well subordinate to the group from being drilled,
       regardless of the availability of other rigs
   -2    This number causes the removal of all drilling rigs currently belonging to the group (but not
       any belonging to subordinate or superior groups).

   DEFAULT: 0

4. Add or remove these rigs
   ADD    Add the rig(s) to the group
   REM    Remove the rig(s) from the group

   DEFAULT: ADD

End the data record with a slash (/).
End the set of records with a blank record, containing just a slash.

If no workover rigs have been specified, a single workover rig (number 100) is automatically be created for
the FIELD group. Similarly, if no drilling rigs have been specified, a single drilling rig (number 100) will
automatically be created for the FIELD group. These default rigs are created for back-compatibility with
pre-95A ECLIPSE 100 and pre-2002A ECLIPSE 300 releases that do not use the GRUPRIG keyword for
both types of rig. They are removed automatically the first time GRUPRIG is used to add a rig to any group.

Rigs may be allocated to groups on different levels. A rig allocated to a group on any level will be available
to all its subordinate wells. For example, let the FIELD contain two groups, PLAT-A and PLAT-B. If rigs 1
and 2 are allocated to the FIELD group, and rig 3 is allocated to group PLAT-A, then all three rigs are
available to the wells subordinate to PLAT-A while only rigs 1 and 2 are available to the wells subordinate
to PLAT-B. Thus, if more than five rigs of a given type are to be available to the whole field, the first five
can be allocated to the FIELD group and the remainder (up to another five) can be allocated to all the
groups on the next level down. ECLIPSE makes no attempt to optimize the use of multiple rigs; to find a
rig for a well it searches up the group hierarchy from the well’s group up to the FIELD and selects the first
free rig that it finds.
The two types of rig operate independently of each other. For example, drilling rig number 1 and workover rig number 1 are separate rigs, and do not affect each other.

**CAUTION:** Pre-96A datasets experience a change in behavior if two or more rigs of either type are allocated to the same group (for example if a group changes its rig during the run). Before the 96A release, the second rig would replace the first one. Now, however, the second rig is added to the group and the first rig still remains available. If you wish to replace one rig with another, you must now remove the old rig then add the new one.

See also keywords **WORKLIM, WDRILTIM, QDRILL, WDRILPRI and PRORDER.**

**Examples**

**Example 1**

Drilling rigs 1 and 2 are allocated to the FIELD. Workover rigs 1 and 2 are allocated to groups PLAT-A and PLAT-B respectively.

<table>
<thead>
<tr>
<th>GRUPRIG</th>
<th>-- group</th>
<th>workover</th>
<th>drilling</th>
<th>add or</th>
</tr>
</thead>
<tbody>
<tr>
<td>-- name</td>
<td>rig</td>
<td>rig</td>
<td>remove</td>
<td></td>
</tr>
<tr>
<td>FIELD</td>
<td>0</td>
<td>1</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>FIELD</td>
<td>0</td>
<td>2</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>PLAT-A</td>
<td>1</td>
<td>0</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>PLAT-B</td>
<td>2</td>
<td>0</td>
<td>/</td>
<td></td>
</tr>
</tbody>
</table>

**Example 2**

Drilling rig 2 is to be removed from the FIELD and just made available to group PLAT-B.

<table>
<thead>
<tr>
<th>GRUPRIG</th>
<th>-- group</th>
<th>workover</th>
<th>drilling</th>
<th>add or</th>
</tr>
</thead>
<tbody>
<tr>
<td>-- name</td>
<td>rig</td>
<td>rig</td>
<td>remove</td>
<td></td>
</tr>
<tr>
<td>FIELD</td>
<td>0</td>
<td>2</td>
<td>REM</td>
<td></td>
</tr>
<tr>
<td>PLAT-B</td>
<td>0</td>
<td>2</td>
<td>ADD</td>
<td></td>
</tr>
</tbody>
</table>

**Keywords**

**1019**
GRUPSALE  Specifies group gas sales rate

This keyword specifies the amount of gas produced from a group that is reserved for sale. This reduces the gas available for reinjection.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Group name or name root
   or FIELD (to set the field sales rate)
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. The constant sales rate. This is summed with the quantity specified using item 3 to give the overall gas sales requirement. A user defined argument can be set with keywords UDQ and UDADIMS.

   UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

   DEFAULT: 0.0

3. The fractional gas sales rate. This is a fraction of the total gas production from the group (before any fuel, sales and reinjection requirements have been taken). This fractional part is calculated at each timestep and then summed with the quantity specified using item 2 to give the overall gas sales requirement. A user defined argument can be set with keywords UDQ and UDADIMS.

   DEFAULT: 0.0

4. The component template
   One value for each component, the default being 1 for all components.

By default, the group fuel requirement is satisfied prior to the sales requirement. This ordering may be changed using the WTAKEGAS keyword.

The treatment of the constant and fractional parts of the sales requirement, and the use of the component template, are the same as that described for fuel under keyword GRUPFUEL.

See also "Group injection control options" in the ECLIPSE Technical Description.

Note: For accurate group gas accounting, there is a limitation that reinjection control and any sales gas or fuel gas requirements must all be specified for the same group, usually the FIELD group, and not split across the group hierarchy. This limitation only applies if the injection gas contains group vapor, with the vapor taken from a separator stage (that is not from the whole separator).

Example

Set group gas sales rate of 1000 Mscf/day.
GRUPSALE
FIELD 1000 /
/
GRUPSLAV  Identifies the slave groups in a slave reservoir

This keyword must be used in each reservoir coupling slave run to identify the slave groups and their associated master groups in the master run, and to set the filter flags that determine which rate constraints are to be imposed on them (see "Reservoir coupling" in the ECLIPSE Technical Description).

If the slave run is activated by the master run (see keyword SLAVES), the slave groups’ rate constraints are transmitted from the master run by the message passing system, at each timestep of the master run.

Alternatively, the slave run may be activated on its own. In this case, the slave groups’ rate constraints are read from the input Reservoir Coupling file, which must be specified with the keyword USECUPL.

In either case, the slave run’s timesteps are synchronized to coincide with the timesteps of the master run (either transmitted from the master run or read from the Reservoir Coupling file). The slave run may, however, take more than one timestep to reach the end of the master run’s timestep.

The GRUPSLAV keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. The slave group’s name or group name root
   or FIELD (if the FIELD group is the slave group)
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record
   The slave reservoir must contain one or more slave groups. They need not all be on the same level in the group tree, but a slave group cannot be subordinate to another slave group. The slave groups must not be subject to production or injection rate constraints applied to any superior group within the slave reservoir.
   The groups must previously have been declared with the keyword GRUPTREE or WELSPECS.

2. The name of the slave group’s associated master group in the master reservoir
   The slave group may have a different name from its master group. For example, the slave group may be the top level group in the slave reservoir, and thus must be called FIELD.
   If a default (1*) or blank name (' ') is entered, ECLIPSE assumes the associated master group has the same name as its slave group.
   In slave runs activated by the master run, ECLIPSE checks that the master-slave group pairs specified in keywords GRUPMAST and GRUPSLAV are consistent.

3. The filter flag for oil production rate constraints (up to 4 characters)
   This determines the source of the oil rate constraints to be applied to the slave group: the master run (or the Reservoir Coupling file), the slave run’s own data file, or both.
   MAST  Apply only the constraints transmitted from the master run or read from the Reservoir Coupling file. Ignore any rate constraints specified for this slave group in the slave run’s own data file.
   SLAV  Apply only the constraints specified for this slave group in the slave run’s own data file. Ignore any rate constraints transmitted from the master run or read from the Reservoir Coupling file.
BOTH  Apply the rate constraints from both sources, taking the minimum of the two values.

DEFAULT: MAST

4. The filter flag for water and liquid production rate constraints
   The choices are the same as for item 3.

5. The filter flag for gas production rate constraints
   The choices are the same as for item 3.

6. The filter flag for reservoir fluid volume production rate constraints
   The choices are the same as for item 3.

7. The filter flag for oil injection rate constraints
   The choices are the same as for item 3.

8. The filter flag for water injection rate constraints
   The choices are the same as for item 3.

9. The filter flag for gas injection rate constraints
   The choices are the same as for item 3.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Notes

• The SLAV option in items 3 - 9 is only intended for use when the slave run is performed on its own
  and reads the slave group rate constraints from the input Reservoir Coupling file. This allows you to
  see how the slave reservoir would perform in the absence of selected rate constraints imposed by the
  master run. The SLAV option should not be used when the slave run is activated by the master run as
  part of the coupled system. Ignoring rate constraints imposed by the master run would cause target or
  limiting rates imposed on the global system to be exceeded.

• If a rate constraint is applied to a phase that is not an active phase in the slave run, ECLIPSE
  transforms it into a constraint on an associated phase, using the latest available production ratios. This
  may happen in the following two circumstances:

  • Gas is an active phase in the master run, but not in the slave run that instead has a constant R_s
    specified with keyword RCONST or RCONSTT.

  • Oil is an active phase in a gas condensate master run, but not in the slave run that instead has a
    constant R_v specified with keyword RVCONST or RVCONSTT.

Examples

Slave groups PL-A and PL-B take their rate constraints from the master run, but in addition PL-B observes
its own limits on water injection rate.

<table>
<thead>
<tr>
<th>GRUPSLAV</th>
</tr>
</thead>
<tbody>
<tr>
<td>slave   assoc oil wat/liq gas resv oil wat gas</td>
</tr>
<tr>
<td>group   master prod prod prod prod inj inj inj</td>
</tr>
<tr>
<td>name    group limits limits limits limits limits limits</td>
</tr>
</tbody>
</table>

Keywords

1023
<table>
<thead>
<tr>
<th>PL-A</th>
<th>UP-A</th>
<th>/</th>
</tr>
</thead>
<tbody>
<tr>
<td>PL-B</td>
<td>UP-B</td>
<td>5*</td>
</tr>
<tr>
<td>/</td>
<td>BOTH</td>
<td>/</td>
</tr>
</tbody>
</table>
GRUPTARG  Resets a group production rate target or limit

This keyword can be used to reset a group production rate target or limit, without the bother of specifying all the other quantities required by the group control keywords GCONPROD or GCONFRI. These other quantities are left unchanged. The group control data should initially have been set using GCONPROD (for guide rate control) or GCONFRI (for prioritization), otherwise guide rate control is applied by default if a limit is exceeded.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root
   or FIELD (for field targets or limits)

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Definition of the target or limit to be changed
   ORAT  Oil rate
   WRAT  Water rate
   GRAT  Gas rate
   LRAT  Liquid rate
   CRAT  Linearly combined rate (coefficients must be set with LINCOM)

   ECLIPSE 300 only  WGRA  Wet gas rate
                    RESV  Reservoir fluid volume rate

   ECLIPSE 300 only  CVAL  Calorific rate
                    PRBL  Reservoir fluid volume production balancing fraction

   ECLIPSE 300 only  PBGS  Surface gas volume production balancing fraction

   ECLIPSE 300 only  PBWS  Surface water volume production balancing fraction

   GUID  Guide rate (see items 9 and 10 of the keyword GCONPROD).

   The guide rate can only be set here if its definition has been set to OIL, WAT, GAS, LIQ, WGA or RES in item 10 of keyword GCONPROD.

3. New value of this quantity.

   The units depend upon the quantity chosen. The appropriate units are listed in the specification for keywords GCONPROD and GCONFRI.

   If an oil, water, gas or liquid rate limit is specified, and the procedure on exceeding the corresponding rate limit has not been set in GCONPROD or GCONFRI, the procedure is defaulted to RATE (for guide rate control) or PRI (for prioritization).
DEFAULT: No target or limit

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also the ECLIPSE 100 keyword `GTMULT`.

**Example**

<table>
<thead>
<tr>
<th>GRUPTARG</th>
</tr>
</thead>
<tbody>
<tr>
<td>PLAT-A  ORAT 15000 /</td>
</tr>
<tr>
<td>PLAT-B  RESV 20000 /</td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>
Sets up tree structure for multi-level group control

This keyword is required only if you need a grouping structure with more than three levels in the hierarchy (that is field - groups - wells). A multi-level tree structure can be built up by using this keyword to specify the immediate parent of each group. The field itself occupies the top of the tree (Level 0). The groups immediately subordinate to the field are on Level 1. Other groups may be placed on Level 2 by giving them a parent on Level 1. Groups can be placed on any lower level in this way.

The keyword is followed by any number of records, each containing two group names and terminated with a slash (/). Group names may have up to 8 characters. Name roots (ending with an asterisk) cannot be used here.

1. The name of the child group
2. The name of its parent group

The record declares the second group to be the parent of the first group.

The set of records must end with a blank record, containing only a slash.

Groups on Level 1 need not have their parent declared, but optionally their parent can be declared as FIELD. Groups without a parent are automatically placed on Level 1.

The group hierarchy can have any number of levels. Groups that have other groups as children cannot also have wells. (Wells are assigned to groups in the keyword WELSPCE.) Thus a group either contains wells (that is a well-group) or has other groups as children (that is a node-group).

The group hierarchy can have any number of levels. However, group controls and guide rates can only be assigned to a maximum of four levels of groups down any branch of the hierarchy, including the FIELD. A group may have both wells and child groups, but this structure is incompatible with that allowed in ECLIPSE 100.

Groups can be reassigned to different parents during the simulation, and new groups can be inserted into the hierarchy, by using GRUPTREE to declare any changes in the parent-child relationships.

If this keyword is omitted, all the groups declared in WELSPCE are assigned to Level 1, giving the standard 3-level field-group-well hierarchy.

Example

To define the hierarchy as shown in figure 3.8:

```
FIELD
  /
  /  
PLAT-A PLAT-B
  /     /
GR-A1  GR-A2  SAT-B  GR-B1  GR-B2
     /   |
    GR-S1 GR-S2
```

Figure 3.8. A five level hierarchy
It is assumed that groups GR-A1, GR-A2, GR-B1, GR-B2, GR-S1 and GR-S2 have wells declared as belonging to them in `WELSPECS`. PLAT-A and PLAT-B can optionally have `FIELD` declared as their parent.

<table>
<thead>
<tr>
<th>GRUPTREE</th>
<th>parent</th>
</tr>
</thead>
<tbody>
<tr>
<td>GR-A1</td>
<td>PLAT-A</td>
</tr>
<tr>
<td>GR-A2</td>
<td>PLAT-A</td>
</tr>
<tr>
<td>GR-B1</td>
<td>PLAT-B</td>
</tr>
<tr>
<td>GR-B2</td>
<td>PLAT-B</td>
</tr>
<tr>
<td>SAT-B</td>
<td>PLAT-B</td>
</tr>
<tr>
<td>GR-S1</td>
<td>SAT-B</td>
</tr>
<tr>
<td>GR-S2</td>
<td>SAT-B</td>
</tr>
</tbody>
</table>
**GSATCOMP Specify a composition for a satellite group**

This keyword specifies the composition of the oil or gas to be injected or produced by a satellite group (see keywords `GSATINJE` and `GSATPROD`).

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. The set of records is ended by a blank record, containing only a slash.

1. Satellite group name or group name root.
   
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record. The group(s) must previously have been declared with keyword `GRUPTREE`, to define their position in the grouping hierarchy.

2. Character string specifying type of composition to be defined.
   
   This must be one of the following:
   
   - `IG` Set composition of injection gas
   - `IO` Set composition of injection oil
   - `PG` Set composition of produced gas
   - `PO` Set composition of produced oil

3. Mnemonic for a well stream defining the composition.
   
   The stream must be defined with the keyword `WELLSTRE`.

4. Oil density value at surface conditions.
   
   This will set the assumed oil density for reporting of surface oil mass rate and mass total. If not specified, it will be estimated from an initial flash of the reservoir fluid.

   **UNITS:** kg/m$^3$ (METRIC), lb/ft$^3$ (FIELD), gm/cc (LAB), kg/m$^3$ (PVT-M)

**Example**

```
GSATCOMP
SG2 IG STR1 /
/ 
```
**Injection rate data for satellite groups**

This keyword specifies the rate at which oil, water and gas is to be injected by a satellite group.

Satellite groups provide a means of incorporating known production and injection flows from other reservoirs or regions that are not part of the simulation grid. They have no associated wells or subordinate groups, but act as sources of production flow and/or sinks of injection flow. The **GSATINJE** keyword is used to specify the injection rates of one or more satellite groups. These rates are held constant until they are reset in a subsequent **GSATINJE** keyword. If the satellite groups are also producing fluid, their production rates must be set with keyword **GSATPROD**. See keyword **GSATPROD** for further description of satellite groups.

When using ECLIPSE 300 for compositional runs, the composition of the injected oil or gas should be defined with keyword **GSATCOMP**. If the fluid compositions are left undefined, the injected oil and gas compositions will default to the compositions obtained by flashing the initial reservoir fluids-in-place through the field separator.

The **GSATINJE** keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. **Satellite group name or group name root**
   
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record. The groups must previously have been declared with keyword **GRUPTREE**, to define their position in the grouping hierarchy.

   If a complete group name (without an asterisk) is entered, and the group has not been declared earlier, ECLIPSE creates a new group with that name and places it on level 1 directly subordinate to the **FIELD**.

   Satellite groups cannot have wells or subordinate groups.

2. **Phase to which the following rates apply**

   **OIL**, **WAT** or **GAS**

   Satellite groups may inject more than one phase. A separate data record is needed for each phase.

3. **Surface injection rate of the phase in item 2**

   The value must be greater than or equal to zero.

   **UNITS**: \(\text{sm}^3/\text{day (METRIC)}\), \(\text{stb/day for oil, water or liquid (FIELD)}\), \(\text{Mscf/day for gas (FIELD)}\), \(\text{scc/hr (LAB)}\), \(\text{sm}^3/\text{day (PVT-M)}\)

   **DEFAULT**: Zero

4. **Reservoir volume injection rate of the phase in item 2**

   You need to enter a value here only if you want the reservoir volume injection rate of the superior groups and the field to include that of the satellite group. If this quantity is left as zero for all the satellite groups, the field’s reservoir volume injection rate will just represent the fluid injected into the simulation grid. The choice is important if a superior group has a reservoir volume injection rate target or a voidage replacement target, or a target production balancing fraction.

   **UNITS**: \(\text{rm}^3/\text{day (METRIC)}\), \(\text{rb/day (FIELD)}\), \(\text{rcc/hr (LAB)}\), \(\text{rm}^3/\text{day (PVT-M)}\)
The value must be greater than or equal to zero.

**ECLIPSE 100 only**

5. Mean calorific value of gas injected by the group

A value need only be entered here if the gas calorific value control option is being used (see "Gas calorific value control" in the *ECLIPSE Technical Description*).

The value must be greater than or equal to zero.

**UNITS:** kJ/sm$^3$ (METRIC), Btu/Mscf (FIELD), J/scc (LAB), kJ/sm$^3$ (PVT-M)

**DEFAULT:** Zero

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

**Example**

```
GSATINJE
   -- group inj surf resv
   -- name phase rate rate
   SAT-A WAT 7520 7520 /
   SAT-A GAS 17760 10240 /
/```

**Keywords**

1031
GSATPROD  Production rate data for satellite groups

This keyword specifies the rate at which oil, water and gas is produced by a satellite group.

Satellite groups provide a means of incorporating known production and injection flows from other reservoirs or regions that are not part of the simulation grid. They have no associated wells or subordinate groups, but act as sources of production flow and/or sinks of injection flow. The GSATPROD keyword is used to specify the production rates of one or more satellite groups. These rates are held constant until they are reset in a subsequent GSATPROD keyword. If the satellite groups are also injecting fluid, their injection rates must be set with keyword GSATINJE.

In ECLIPSE 300. In compositional runs, the composition of the produced hydrocarbon fluids should be defined with keyword GSATCOMP. If the fluid compositions are left undefined, the produced oil and gas compositions default to the compositions obtained by flashing the initial reservoir fluids-in-place through the field separator.

The group control facilities will take account of flows to and from satellite groups. If a superior group has a rate target, the flows from satellite groups, and any other groups declared unavailable for control from a higher level, will first be determined. Then the other subordinate groups or wells make up the remainder of the flow target, if they have sufficient potential.

The first time a group is referenced in either keyword GSATPROD or GSATINJE it is defined to be a satellite group. In a multilevel group hierarchy, the satellite group should previously be declared in keyword GRUPTREE to define its position in the grouping hierarchy, otherwise by default the group is placed on level 1 immediately subordinate to the FIELD. An error message is issued if the group has already been given wells or subordinate groups. Once defined as a satellite group, only keywords GSATPROD and GSATINJE should be used to set its flow rates; GCONPROD and GCONINJE should not be used.

When running the network option, satellite groups can be included in the production or injection network with the keyword GRUPNET or GNETINJE respectively. Nodal pressures will be calculated for these groups, as for all other groups in the network, but of course the satellite groups' flow rates are fixed at their specified values and do not depend on their nodal pressures. When running the extended network model, satellite groups are source groups and must therefore have a network node with the same name; they are included in the network with the keywords NODEPROP and BRANPROP. When running the gas field operations model, satellite groups cannot be made contract groups.

The GSATPROD keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Satellite group name or group name root

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record. The groups must previously have been declared with keyword GRUPTREE, to define their position in the grouping hierarchy.

   If a complete group name (without an asterisk) is entered, and the group has not been declared earlier, ECLIPSE creates a new group with that name and places it on level 1 directly subordinate to the FIELD.

   Satellite groups cannot have wells or subordinate groups.

2. Oil production rate from the group

   The value must be greater than or equal to zero.
3. Water production rate from the group
   The value must be greater than or equal to zero.
   UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)
   DEFAULT: Zero

4. Gas production rate from the group
   The value must be greater than or equal to zero.
   UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)
   DEFAULT: Zero

5. Reservoir fluid volume production rate from the group
   The value must be greater than or equal to zero.
   You need to enter a value here only if you want the reservoir fluid volume rate of the superior groups and the field to include that from the satellite group. If this quantity is left as zero for all the satellite groups, the field's reservoir fluid volume rate will just represent the fluid produced from the simulation grid. The choice is important if a superior group has a reservoir fluid volume production rate target or production balancing fraction, or a voidage replacement injection target.
   UNITS: rm$^3$/day (METRIC), rb/day (FIELD), rcc/hr (LAB), rm$^3$/day (PVT-M)
   DEFAULT: Zero

6. Lift gas supply rate to the group
   You need enter a value here only if lift gas is being supplied to the group and you are running the Network option (ECLIPSE 100 only) or the Gas Lift Optimization facility.
   The lift gas is added to the production network flows, for the purpose of calculating the pressure drop along each network branch, in branches that have item 6 of the GRUPNET keyword set to FLO, or in extended network nodes that have item 4 of keyword NODEPROP set to YES (see the "Network option" in the ECLIPSE Technical Description).
   When running the Gas Lift Optimization facility (see the "Gas lift optimization" in the ECLIPSE Technical Description), the group's lift gas supply rate is accounted against any superior group's lift gas supply limit set with keyword GLIFTOPT.
   The value must be greater than or equal to zero.
   UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)
   DEFAULT: Zero

7. Mean calorific value of gas produced by the group
   A value need only be entered here if the gas calorific value control option is being used (see "Gas calorific value control" in the ECLIPSE Technical Description).
   The value must be greater than or equal to zero.
   UNITS: kJ/sm$^3$ (METRIC), Btu/Mscf (FIELD), J/scc (LAB), kJ/sm$^3$ (PVT-M)
DEFAULT: Zero

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Notes

ECLIPSE 300 only

• Separators only act on the component molar rates produced from wells; the resultant phase surface volume rates from each well in a group are then summed to form the group's phase surface volume rates. This summation is also done for the gas volume rates at each stage of the separator to give group level stage gas volume rates. Satellite groups do not have any wells so, for reporting purposes, the satellite group stage 1 gas volume rate is set to the group gas production rate (so that the group rate and the sum of the group stage rates match).

• In complex separator chains, where the stage 1 separator gas is used as a feed further down the separator chain, the setting of the satellite group stage 1 gas volume rate to the group gas production rate may cause confusion, as it is not added to the latter feed. Item 113 in OPTIONS3 can be used to turn off the setting of satellite group stage 1 gas volume rate to the group gas production rate to avoid such confusion.

Example

```
GSATPROD
-- group     oil    water    gas     void    lift
-- name      rate   rate     rate     rate    gas
SAT-A  14500   2640    17600   /        /
SAT-B  12700   3270    15800   /        /
```
**GSEPCOND**  Assigns separators to groups in ECLIPSE 100

This keyword is used to associate each separator, previously named in keyword `SEPVALS`, with a particular group. It is required if you wish to model the effects of changing separator conditions in ECLIPSE 100 runs. If a separator is assigned to a particular group, all wells subordinate to that group use that separator. If the separator’s conditions are subsequently altered by redefining them with `SEPVALS`, the oil and gas flow rates of these wells are transformed to reflect the change in conditions. See keyword `SEPVALS` for a more detailed discussion on changing separator conditions.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root  or FIELD (if the separator is at the field level)

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Name of the separator associated with the group

3. (and its subordinate groups and wells).

   The separator named here must already have been specified in keyword `SEPVALS`.

   A blank name (" ") removes any previous association between the group and a separator.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

**Examples**

**Example 1**

To assign a separator to the whole field:

```
GSEPCOND
FIELD  SEP1 /
/
```

**Example 2**

To assign different separators to two platforms:

```
GSEPCOND
PLAT-A  SEP1 /
PLAT-B  SEP2 /
/
```
Assign separator conditions to a group for group flash summary vectors

This keyword sets the separator, EoS and PVT region numbers for the flash calculation used for group level flash summary keywords. These keywords are listed in table 2.25 and end in $G$.

**Note:** If any of the summary section keywords ending in $G$ are used then warnings will be issued for any groups with ambiguous equation of state or PVT region numbers. To eliminate these warning messages add a new GSEPREPT entry for the group which incurred the warning.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated at any item after item 1. The remaining items assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root
   
   or FIELD (for field level flash calculation)

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Separator name. This must either be ’FIELDSEP’ or be defined by SEPCOND.
   
   DEFAULT: The first separator assigned to a parent group by SEPCOND will be used.

3. The surface equation of state number.

   This should only be specified if multiple equations of state are used. When this option is employed it may still be defaulted, in which case the fluid from a group is flashed using the equation of state used for the first well assigned to this group. If this group has no wells then the equation of state used by the first well assigned to the first group assigned to this group is used, followed by the first well assigned to the second group assigned to this group and so on through the group tree until such a well exists.

   **Note:** A warning is given if wells contributing to this group use more than one equation of state. If such a warning is given it is not meaningful to compare the group level flash to the sum of the well level flashes.

4. The PVT table number.

   This should be specified if multiple PVT regions are specified. When this option is employed it may still be defaulted, in which case the fluid from a group is flashed using the PVT table number used for the first well assigned to this group. If this group has no wells then the PVT table number assigned to the first well assigned to the first group assigned to this group is used, followed by the first well
assigned to the second group assigned to this group and so forth through the group tree until such a
well exists.

**Note:** A warning is given if wells contributing to this group use more than one PVT table number. If such a
warning is given it is not meaningful to compare the group level flash to the sum of the well level flashes.

### Example

```
GSEPRESPT
  PLAT-A FIELDSEP 1 1 /
  PLAT-B SEP1 2 1 /
/```

**Keywords**

1037
**Gas saturation functions (GASWAT/CO2STORE cases)**

This keyword is used in conjunction with the `WSF` keyword in GASWAT or CO2STORE (non-thermal) cases.

The data specified consists of a number of tables of gas saturation data, each terminated with a slash (/). Each table consists of 3 columns of data. The number of tables should be equal to be number of saturation tables in the run, specified by the `TABDIMS` keyword, and defaulting to 1.

Column:

1. The gas saturation.
   Values should be between 0 and 1 and should increase monotonically down the column.

2. The corresponding gas relative permeability.
   Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0.

3. The corresponding gas-water capillary pressure.
   Values should be level or increasing down the column.
   
   **UNITS:** bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

There must be the same number of entries in each column of a given table, and the number of rows should be less than or equal to the maximum number of rows of saturation data specified with `TABDIMS`.

**Example**

```
GSF
.0000 .0000 .0000
.0400 .0000 .2000
.1000 .0220 .5000
.2000 .1000 1.0000
.4000 .3400 2.0000
.6000 .5000 3.0000
.7800 1.0000 3.9000
/```

ECLIPSE Reference Manual

**Keywords**

1038
GSSCPTST  **Performs a sustainable capacity test**

This keyword causes a sustainable capacity test to be carried out. The model is saved, and then run forward to establish whether a group of wells could sustain a certain gas or oil production rate for a certain period of time. Depending on the options selected, the test will then try to establish the maximum possible production rate sustainable for the test period by attempting to simulate the test period with different group production rates. After the test has been completed ECLIPSE will run on from the reservoir state before the test began.

The GSSCPTST keyword should be followed by the items described below, terminated with a slash (/):

1. **Group name or group name root or FIELD (for field rate control)**
   - **DEFAULT:** FIELD

2. **Group production rate control mode**
   - The group of wells may be placed under the control modes listed below
     - GRAT  Gas rate control
     - ORAT  Oil rate control
     - WRAT  Water rate control
   - **ECLIPSE 300 only**
     - WGRA  Wet gas rate control
   - **DEFAULT:** GRAT

3. **Target production rate**
   - **UNITS:** sm$^3$/day (**METRIC**), stb/day for oil or water (**FIELD**), Mcf/day for gas (**FIELD**), scf/hr (**LAB**), sm$^3$/day (**PVT-M**)
   - **DEFAULT:** No production

4. **Target production period**
   - **UNITS:** day (**METRIC**), day (**FIELD**), hr (**LAB**), day (**PVT-M**)
   - **DEFAULT:** No period

5. **Maximum production rate calculation flag**
   - 1 Run a single iteration at the target production rate until either the target period has been achieved or the group cannot sustain the production rate.
   - 2 As for 1, but if target rate is not sustainable run further iterations to find the maximum rate which is sustainable for the target production period.
   - 3 Run a number of iterations to try to find the maximum rate sustainable for the target production period. For this option, item 3 serves only as a first guess.
   - **DEFAULT:** Option 1

6. **Convergence tolerance**

---

**Keywords**

<table>
<thead>
<tr>
<th>ECLIPSE 100</th>
<th>x</th>
</tr>
</thead>
<tbody>
<tr>
<td>SPECIAL</td>
<td></td>
</tr>
<tr>
<td>RUNSPEC</td>
<td></td>
</tr>
<tr>
<td>GRID</td>
<td></td>
</tr>
<tr>
<td>EDIT</td>
<td></td>
</tr>
<tr>
<td>PROPS</td>
<td></td>
</tr>
<tr>
<td>REGIONS</td>
<td></td>
</tr>
<tr>
<td>SOLUTION</td>
<td></td>
</tr>
<tr>
<td>SUMMARY</td>
<td></td>
</tr>
<tr>
<td>SCHEDULE</td>
<td>x</td>
</tr>
</tbody>
</table>
If item 5 is set to 2 or 3, stop searching for a maximum sustainable rate if the maximum sustainable rate and minimum unsustainable rate are separated by this tolerance. A more generous tolerance will result in less time being spent on the test, but the result will be less informative. The maximum sustainable rate is the greatest production rate tested which can be maintained for the test period. The minimum unsustainable rate is the lowest production rate tested which cannot be maintained for the test period.

UNITS: sm³/day (METRIC), stb/day for oil or water (FIELD), Mcf/day for gas (FIELD), scc/hr (LAB), sm³/day (PVT-M)

DEFAULT: No tolerance (use all iterations)

7. Maximum number of iterations

If item 5 is set to 2 or 3, stop searching for a maximum sustainable rate after this many iterations.

DEFAULT: 6 iterations

8. Subgroup control flag

The control of subgroups of the group named in element 1 will be set to GRUP. This flag specifies changes to rate limits for subgroups

0  No change to subgroup rate limits.
1  Any subgroup gas rate limits removed.
2  As option 1, but wet gas and calorific rate limits are also removed.
3  As option 2, but liquid, oil, water and voidage production rate limits are also removed.

DEFAULT: Option 0

The results of the sustainable capacity test are printed in the PRT file. SUMMARY section keywords FGSTP, FGSPR, FGSSP, FGSRU, FGSR, GGSTP, GGSPR, GGSRU and GGSR do all give information on the most recent sustainable capacity test (SCT) for gas. The SUMMARY section keywords FOSTP, FOSPR, FOSSP, FOSRU, FOSR, GOSTP, GOSPR, GOSR, GOSR and GOS do the same for an oil SCT. Similarly replacing O with W in these keywords provides the SCT for water.

If the user is interested in the accuracy of the time period outputs of GSSCPTST (for example FGSSP) then it may be advisable to set TSMAXZ using TUNING before any GSSCPTST calls. The time period outputs are only as accurate as the maximum timestep used. It may also be useful to increase NUPCOL to ensure that the well system converges in one timestep.

The first iteration will use item 3 as the target production rate for the nominated group. Subsequent iterations use the following algorithm to set the target rate:

- If no unsustainable rate has been found, double previous rate.
- If no sustainable rate has been found, halve previous rate.
- If only one unsustainable rate has been found, bisect the sustainable rate and the lowest unsustainable rate
- If two unsustainable rates have been found the following formula is used \( R = U_0 + \frac{(U_0 - U_1)(T_f - T_0)}{T_0 - T_1} \)

where
\( R \) is the new rate
\( U_0 \) is the previous minimum unsustainable rate
\( U_1 \) is the previous next-to-minimum unsustainable rate
\( T_0 \) is the time when rate \( U_0 \) became unsustainable
\( T_1 \) is the time when rate \( U_1 \) became unsustainable

and \( T_f \) is the time of the end of the sustained capacity test.

If the final method suggests a rate which is too high then the rate is set to \( R = U_0 - Tol \) where \( Tol \) is the tolerance specified in item 6. If \( T_0 = T_1 \) then a bisection is used.

**Example**

To see if group GP can produce gas at 50000 Mscf/day for a year use the following (assuming FIELD units):

```
GSSCPTST
GP GRAT 50000.0 365.25 1 /
```

To see what gas rate could be maintained for a decade (to within 100 Mscf and again assuming FIELD units), use the following:

```
GSSCPTST
GP GRAT 50000.0 3652.5 3 100.0 12 /
```

**Note:** This second example could require up to 120 years of simulated reservoir time, and is therefore likely to take some time to complete.
Swing and profile factors for multiple contract groups

The GSWINGF keyword is used in place of SWINGFAC to turn on the Gas Field Operations Model in the Multiple Contract Groups mode. This allows two or more separate groups to have their own independent gas supply contracts. Each contract group has its own set of swing factors and profile factors, and its own DCQ value which ECLIPSE will adjust independently to meet the swing requirement if requested to do so in the GASYEAR, GASPERIO and GDCQ keywords.

The GSWINGF keyword is used to identify the contract groups and supply their swing and profile factors. The following features of the Gas Field Operations Model are also activated when either GSWINGF or SWINGFAC is first entered:

- Group and field gas rate targets are applied to sales gas rates instead of gross gas production rates.
- The delivery capacity is calculated at the beginning of each timestep.
- In ECLIPSE 100, any automatic compressors (specified with keyword GASFCOMP or NETCOMPA) will be turned on and off as required to meet the group or field gas targets.

The swing factors and profile factors themselves are used to set the group gas production targets and to adjust their DCQs when the simulation is advanced using the keyword GASYEAR or GASPERIO instead of DATES, TIME or TSTEP. Each contract group is automatically placed under Guide Rate Control to meet any gas rate targets imposed, unless Prioritization is requested with the keyword GCONPRI.

In order to use the Multiple Contract Groups facility, item 1 of the GASFIELD keyword in the RUNSPEC section must be set to YES.

The set of contract groups must obey the following restrictions:

- One contract group must not be subordinate to another contract group.
- The set of contract groups must cover the whole field, that is every production well must be subordinate to a contract group.
- There must be no gas consumption or import at any node downstream of a contract group (otherwise the contract group’s sales gas targets would not be set correctly).

See "Applying separate gas supply contracts to individual groups" in the ECLIPSE Technical Description for further details.

The GSWINGF keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1 Name or name root of the Contract Group(s) to which the following swing and profile factors apply
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2 - 13 Swing factors for each month in sequence from January to December.
   Setting January’s swing factor to a negative value causes the group to stop being a contract group.

14 - 25 Profile factors for each month in sequence from January to December.
A warning is printed if the profile factor for any month is greater than that month’s swing factor. If this is so, obeying the swing requirement may not guarantee the group could produce its target rate profile.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

**Normalized profile factors**

In order to produce the required ACQ for a given DCQ, the profile factors should be normalized such that

- the sum over all months of \( \text{profile factor} \times \text{number of days in month} \) = 365.25,
- with February allocated 28.25 days.

If this “normalization requirement” is obeyed, then when leap years are taken into account every fourth year, the average annual production at a constant DCQ will be \( DCQ \times 365.25 \), which is the ACQ. A warning is printed if the profile factors are not normalized.

**Example**

<table>
<thead>
<tr>
<th>GROUP</th>
<th>JAN</th>
<th>FEB</th>
<th>MAR</th>
<th>APR</th>
<th>MAY</th>
<th>JUN</th>
<th>JUL</th>
<th>AUG</th>
<th>SEP</th>
<th>OCT</th>
<th>NOV</th>
<th>DEC</th>
</tr>
</thead>
<tbody>
<tr>
<td>PLAT-A</td>
<td>2*1.5036</td>
<td>3*1.0024</td>
<td>3*0.5012</td>
<td>3*1.0024</td>
<td>1.5036</td>
<td>2*1.5036</td>
<td>3*1.0024</td>
<td>3*0.5012</td>
<td>3*1.0024</td>
<td>1.5036</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>PLAT-B</td>
<td>2*1.6551</td>
<td>3*1.0031</td>
<td>3*0.3511</td>
<td>3*1.0031</td>
<td>1.6551</td>
<td>2*1.6551</td>
<td>3*1.0031</td>
<td>3*0.3511</td>
<td>3*1.0031</td>
<td>1.6551</td>
<td>/</td>
<td></td>
</tr>
</tbody>
</table>
GTADD

Adds to a group rate target or limit

This keyword can be used to add to a group production rate, or injection rate target or limit, by a specified amount. The group control data with the initial target or limit should first have been entered using GCONPROD (for guide rate production control), GCONINJE (for guide rate injection control) or GCONPRI (for prioritization).

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root
   or FIELD (for field targets and limits)

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Definitions of the target or limit to be changed are listed below.

   This item can be a user defined quantity (UDA) and can be set with the keywords UDQ and UDADIMS. The values in the brackets are the corresponding values to identify the control quantity if UDA is used for this item.

   ORAT [1] Oil surface production rate
   GRAT [3] Gas surface production rate
   LRAT [4] Liquid surface production rate
   CRAT [5] Linearly combined rate (coefficients must be set with LINCOM)
   RESV [6] Reservoir fluid volume production rate
   PRBL [7] Reservoir fluid volume production balancing fraction
   GUID [8] Production guide rate (see items 9 and 10 of the keyword GCONPROD).

   The guide rate can only be modified here if its definition has been set to OIL, WAT, GAS, LIQ or RES in item 10 of keyword GCONPROD.

   WINJ [9] Water surface injection rate
   GINJ [10] Gas surface injection rate
   GINR [12] Gas reservoir volume injection rate

   *ECLIPSE 100 only* OINJ [13] Oil surface injection rate
   *ECLIPSE 100 only* OINR [14] Oil reservoir volume injection rate
ECLIPSE 300 only  PBGS [13] Surface gas volume production balancing fraction

ECLIPSE 300 only  PBWS [14] Surface water volume production balancing fraction

ECLIPSE 300 only  CVAL [15] Calorific production rate.

Note that if the UDQ for this item starts with 'GUID', then the production guide rate control option will be switched on.

3. Amount to add for this quantity. This quantity can be a UDA and can be set with the keywords UDQ and UDADIMS.

   UNITS: the units depend upon the quantity chosen. Negative numbers are allowed. The appropriate units are listed in the documentation for keyword GCONPROD, GCONINJE and GCONPRI.

4. Number of times addition will be performed. Note that addition will be performed at report timesteps only and this item is active only when a UDA is used for item 3. This item will be truncated to the greatest integer lower than or equal to its value. Also note that any value less than 1 will be defaulted to 1.

   DEFAULT: 1

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Example

```
GTADD
PLAT-A ORAT -500 /
PLAT-B RESV 80000/
/ 
```

Keywords

1045
GTMULT

**Multiplies a group rate target or limit**

This keyword can be used to multiply a group production rate or injection rate target or limit by a specified factor. The group control data with the initial target or limit should first have been entered using \texttt{GCONPROD} (for guide rate production control), \texttt{GCONINJE} (for guide rate injection control) or \texttt{GCONPRI} (for prioritization).

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Group name or group name root
   
or \texttt{FIELD} (for field targets and limits)
   
   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

2. Definitions of the target or limit to be changed are listed below. This item can be a user defined quantity (UDA) and can be set with the keywords \texttt{UDQ} and \texttt{UDADIMS}. The values in the brackets are the corresponding values to identify the control quantity if UDA is used for this item.

   - \texttt{ORAT [1]} Oil surface production rate
   - \texttt{WRAT [2]} Water surface production rate
   - \texttt{GRAT [3]} Gas surface production rate
   - \texttt{LRAT [4]} Liquid surface production rate
   - \texttt{CRAT [5]} Linearly combined rate (coefficients must be set with \texttt{LINCOM})
   - \texttt{RESV [6]} Reservoir fluid volume production rate
   - \texttt{PRBL [7]} Reservoir fluid volume production balancing fraction
   - \texttt{GUID [8]} Production guide rate (see items 9 and 10 of the keyword \texttt{GCONPROD}).
     The guide rate can only be modified here if its definition has been set to \texttt{OIL}, \texttt{WAT}, \texttt{GAS}, \texttt{LIQ} or \texttt{RES} in item 10 of keyword \texttt{GCONPROD}
   - \texttt{WINJ [9]} Water surface injection rate
   - \texttt{GINJ [10]} Gas surface injection rate
   - \texttt{WINR [11]} Water reservoir volume injection rate
   - \texttt{GINR [12]} Gas reservoir volume injection rate
   - \texttt{OINJ [13]} Oil surface injection rate
   - \texttt{OINR [14]} Oil reservoir volume injection rate
   - \texttt{PBGS [13]} Surface gas volume production balancing fraction

\textbf{ECLIPSE 100 only}

- \texttt{PBGS [13]} Surface gas volume production balancing fraction
**ECLIPSE Reference Manual**

_ECLIPSE 300 only_  
PBWS [14] Surface water volume production balancing fraction

**ECLIPSE 300 only**  

Note that if the UDQ for this item starts with 'GUID', then the production guide rate control option will be switched on.

3. Multiplying factor for this quantity. This quantity can be a UDA and can be set with the keywords UDQ and UDADIMS.

   UNITS: dimensionless

4. Number of times multiplication will be performed. Note that multiplication will be performed at report timesteps only and this item is active only when a UDA is used for item 3. This item will be truncated to the greatest integer lower than or equal to its value. Also note that any value less than 1 will be defaulted to 1.

   DEFAULT: 1

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

**Example**

<table>
<thead>
<tr>
<th>GTMULT</th>
</tr>
</thead>
<tbody>
<tr>
<td>PLAT-A   ORAT 1.5 /</td>
</tr>
<tr>
<td>PLAT-B   RESV 0.8 /</td>
</tr>
</tbody>
</table>

Keywords

1047
GUIDECAL  Scales guide rates according to calorific value

This keyword provides an additional way of influencing the mean calorific value of the produced gas, for use with the gas calorific value control option (see "Gas calorific value control" in the ECLIPSE Technical Description), by calculating the individual well guide rates (and any group guide rates) as a function of their calorific values. Well guide rates that are not set directly with keyword WGRUPCON are calculated at the start of each timestep as a function of their production potentials. By default the guide rates are set equal to their production potentials, but the keyword GUIDERAT can be used to specify the coefficients of a more general function of production potentials.

The keyword GUIDECAL can be used to make the guide rate calculation take account of calorific values also. It sets the coefficients A and B of a function $B_W$ which depends on the calorific value $C_{gW}$

$$B_W = (1 + A C_{gW})^B$$

Eq. 3.77

When a well W has its guide rate calculated from its production potentials (that is not set to a fixed value with keyword WGRUPCON), the resulting guide rate is multiplied by $B_W$. Thus by setting appropriate values for the coefficients A and B in keyword GUIDECAL, it is possible to either increase or decrease the well guide rates with increasing calorific value. For example, setting a positive value for B bias the production towards wells with higher calorific values, while negative values of B bias the production towards wells with lower calorific values.

If groups are given guide rates that are calculated from the guide rate formula in keyword GUIDERAT (that is item 10 in GCONPROD is set to FORM), their guide rates are also multiplied by $B_G$, which is calculated as for $B_W$ but using the groups’ mean calorific values.

The GUIDECAL keyword requires two items of data, terminated with a slash (/):

1. The coefficient A in equation 3.77
   This must not be negative.
2. The coefficient B in equation 3.77

Example

```
GUIDECAL
0.001  1.0 /
```
GUIDERAT  

Specifies general formula for guide rates

This keyword can be used to specify a general formula for calculating production guide rates used for group control (see keyword GCONPROD). Subsequently, the formula is used to calculate the production guide rates for:

- All production wells that do not have fixed guide rates set with keyword WGRUPCON, and
- Any group whose guide rate definition is set to 'FORM' in item 10 of keyword GCONPROD.

Group flow rate targets are apportioned between the individual wells in proportion to their guide rates (see "Group production rate targets - guide rate control" in the ECLIPSE Technical Description). By default, each well has its guide rate set equal to its potential flow rate (that is what it could flow immediately in the absence of any rate constraints, at the current grid block conditions) at the beginning of each timestep, subject to limits defined in the keyword DGRDT for ECLIPSE 300. But it could be advantageous to weight the guide rates to discriminate against wells with high GOR or water cut. This keyword provides a means of automatically weighting production well guide rates to take account of their production ratios.

Groups may also be given guide rates, but only if they are required to produce a particular proportion of a rate target applied to a superior group. The formula in this keyword can optionally be applied to calculate production guide rates for individual groups as a function of their potential production rates, if requested in item 10 of keyword GCONPROD. Group potentials are the sum of the potentials of their subordinate open wells.

At the beginning of the next timestep, the guide rates (GR\(_p\)) for a nominated phase \(p\) is calculated from the formula

\[
GR_p = \frac{\left(POT_p\right)^A}{B + C(R_1)^D + E(R_2)^F}
\]

Eq. 3.78

where

- \(POT_p\) is the well’s or group’s potential flow rate of the nominated phase \(p\) (see "Well potentials" in the ECLIPSE Technical Description)
- \(A, B, C, D, E\) and \(F\) are user-supplied powers and coefficients
- \(R_1\) and \(R_2\) are ratios of phase potentials, which depend on the nominated phase \(p\) (see item 2).

For the rest of the simulation, the guide rates are recalculated at the beginning of each timestep that starts after a specified interval (item 1) has elapsed from the previous guide rate calculation. The interval defaults to zero, giving a guide rate calculation at the beginning of each timestep. However, a longer interval may be preferred if the guide rates oscillate due to rate-dependent coning, or alternatively a damping factor may be applied in item 10.

The denominator is not allowed to become negative. If the power \(D\) or \(F\) is negative, the ratio \(R_1\) or \(R_2\) is limited to be not less than 1.0E-6.

If the nominated phase \(p\) differs from the phase under group control, GR\(_p\) is converted into a guide rate for the controlled phase using the well’s or group’s production ratios at the beginning of the timestep.

The GUIDERAT keyword is followed by a line containing some or all of the following data items, terminated with a slash (/).

The items are:
1. The minimum time interval between guide rate calculations.
   The guide rates are calculated from the formula above at the beginning of each timestep that starts after the specified interval has elapsed since the previous guide rate calculation.
   A zero value causes the guide rates to be recalculated every timestep. A longer interval may be preferred if the guide rates oscillate due to rate-dependent coning phenomena, or alternatively a damping factor may be applied in item 10.
   **UNITS:** day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
   **DEFAULT:** 0.0

2. The nominated phase \( p \)
   - **OIL** \( \text{GR}_p \) is the oil phase guide rate.
   - The phase potential ratios are:
     - \( R_1 = \) water-oil ratio (dimensionless)
     - \( R_2 = \) gas-oil ratio
   - **UNITS:** \( \text{sm}^3/\text{sm}^3 \) (METRIC), Mscf/stb (FIELD), scc/scc (LAB), \( \text{sm}^3/\text{sm}^3 \) (PVT-M)
   - **LIQ** \( \text{GR}_p \) is the liquid phase guide rate (oil + water).
   - The phase potential ratios are:
     - \( R_1 = \) water cut (dimensionless)
     - \( R_2 = \) gas-liquid ratio
   - **UNITS:** \( \text{sm}^3/\text{sm}^3 \) (METRIC), Mscf/stb (FIELD), scc/scc (LAB), \( \text{sm}^3/\text{sm}^3 \) (PVT-M)
   - **GAS** \( \text{GR}_p \) is the gas phase guide rate.
   - The phase potential ratios are:
     - \( R_1 = \) water-gas ratio
     - \( R_2 = \) oil-gas ratio
   - **UNITS:** \( \text{sm}^3/\text{sm}^3 \) (METRIC), stb/Mscf (FIELD), scc/scc (LAB), \( \text{sm}^3/\text{sm}^3 \) (PVT-M)
   - **RES** \( \text{GR}_p \) is the reservoir fluid volume guide rate.
   - The phase potential ratios are:
     - \( R_1 = \) water-oil ratio (dimensionless)
     - \( R_2 = \) gas-oil ratio
   - **UNITS:** \( \text{sm}^3/\text{sm}^3 \) (METRIC), Mscf/stb (FIELD), scc/scc (LAB), \( \text{sm}^3/\text{sm}^3 \) (PVT-M)
   - **COMB** \( \text{GR}_p \) is the linearly combined phase guide rate (coefficients must be set with LINCOM).
   - The phase potential ratios are:
     - \( R_1 = \) water to linearly combined phase ratio (dimensionless)
UNITS: $\text{sm}^3/U \text{ (METRIC)}$, $\text{stb}/U \text{ (FIELD)}$, $\text{scc}/U \text{ (LAB)}$, $\text{sm}^3/U \text{ (PVT-M)}$ where $U$ represents the units specified in LCUNIT.

$R_2 = \text{gas to linearly combined phase ratio}$

UNITS: $\text{sm}^3/U \text{ (METRIC)}$, $\text{Mscf}/U \text{ (FIELD)}$, $\text{scc}/U \text{ (LAB)}$, $\text{sm}^3/U \text{ (PVT-M)}$ where $U$ represents the units specified in LCUNIT.

NONE  The general formula is not applied.

The well guide rates is set equal to their production potentials at every timestep, with no damping, just as if this keyword had not been entered. An error is flagged if any group requires its guide rate to be calculated from the formula.

DEFAULT: NONE.

3. The power $A$ of the phase potential rate.
   This must lie between 3.0 and -3.0.
   DEFAULT: 0.0

4. The constant $B$ on the denominator.
   This must not be negative.
   DEFAULT: 0.0

5. The coefficient $C$ of the phase ratio $R_1$.
   DEFAULT: 0.0

6. The power $D$ of the phase ratio $R_1$.
   This must lie between 3.0 and -3.0.
   DEFAULT: 0.0

7. The coefficient $E$ of the phase ratio $R_2$.
   DEFAULT: 0.0

8. The power $F$ of the phase ratio $R_2$.
   This must lie between 3.0 and -3.0.
   DEFAULT: 0.0

9. Should guide rates be allowed to increase?
   
   YES  Guide rates are allowed to increase if the formula results in a higher value (for example due to a reduced water cut or GOR).

   NO  The guide rate of the nominated phase is only allowed to decrease. If the formula results in a higher value than the current one, the value remains unchanged. (This option decreases the tendency for guide rates to oscillate due to rate-dependent coning phenomena.)

   DEFAULT: YES

10. Damping factor, GRDAMP
This must lie between 0.0 and 1.0. It provides a means (as an alternative to setting item 9 to NO) of damping guide rate oscillations that may result from rate-dependent water cut or GOR. Each time a new guide rate is calculated for a well or group, it is averaged with its previous value according to the formula

\[ GR_{\text{used}} = GR_{\text{DAMP}} \times GR_{\text{calculated}} + (1-GR_{\text{DAMP}}) \times GR_{\text{previous}} \]  

Eq. 3.79

A damping factor of 1.0 allows the guide rates to change instantaneously with the phase potentials. As the damping factor reduces towards 0.0, the guide rates will be weighted increasingly towards their previous values, reducing any tendency to oscillate.

DEFAULT: 1.0

**ECLIPSE 100 only**  
11. Use free gas and not total gas potential rates in the phase potential ratio \( R_2 \).

**YES** Only use free gas (at well-block conditions) in the potential ratio.

**NO** Use total (free and dissolved) gas in the potential ratio.

DEFAULT: NO

**ECLIPSE 300 only**  
12. Minimum guide rate \( GR_{\text{min}} \). If the calculated guide rate \( GR_p \) is less than the minimum \( GR_{\text{min}} \), it will be set to the minimum. Very small guide rates may cause a group to be ignored, particularly if the group flow is large. Setting a minimum value prevents this. The minimum should be chosen to ensure

\[ GR_{\text{min}} > 10^{-19} \times \text{flow} \]

DEFAULT: \( 10^{-6} \)

In the absence of keyword GUIDERAT, the well guide rates are set equal to the wells’ potential rates at the beginning of each timestep (unless fixed values have been supplied with keyword WGRUPCON).

The keyword affects production guide rates only. Guide rates for injection wells are still by default set equal to their potential injection rates.

**Examples**

**Example 1**

To set the liquid phase guide rates equal to the liquid phase potentials at low water cut, and biased against high water cut wells, with a damping factor of 0.1:

```
GUIDERAT -- interval phase A B C D E F increase? damp
1*       LIQ 1.0 1.0 10.0 2.0 0.0 0.0     NO        0.1 /
```

**Example 2**

To set the oil phase guide rates equal to the oil phase potentials, to be recalculated after 100 day intervals:

```
GUIDERAT -- interval phase A B C D E F increase? damp
100      OIL 1.0 1.0      /       
```
GUPFREQ  Update frequency for the instantaneous gradient option

This keyword specifies the update frequency of the instantaneous gradient option, see the "Instantaneous gradients" section in the ECLIPSE Technical Description. In contrast to the history matching gradients the instantaneous gradients can be calculated at intermittent timesteps.

1. Update frequency.
   This must be one of:

   ALL  Instantaneous gradients calculated at every subsequent timestep.

   NEXT Instantaneous gradients calculated at the next timestep only. Gradient data frozen until next update requested.

   DEFAULT: ALL

Example

```
GUPFREQ
NEXT / `
GWRTWCV  Instantaneous gradients with respect to well control variables

This keyword specifies collections of wells corresponding to instantaneous gradient parameters, see the "Instantaneous gradients" section in the ECLIPSE Technical Description. ECLIPSE will proceed to compute the instantaneous gradients of particular well properties with respect to the well control variables in the associated parameters. The gradient data is available using the summary keywords WOFWCn, WWFWCn, WGFWCn and WBHWCn, where n is the parameter number requested.

Note that the gradient data is instantaneous in the sense that the derivatives approximate the change in well properties at the current simulation time with respect to a variation in the well control variable at the current simulation time. In contrast, the history matching approach generates gradient data at the current simulation time with respect to variations at the initial time. The gradients calculated are due to the response of the reservoir rather than the well group hierarchy and therefore the wells considered must be under either oil, water, gas rate or bottom hole pressure control.

The maximum number of gradient parameters is specified by the GDIMS keyword in the RUNSPEC section. The instantaneous gradient option can be activated and deactivated during the course of a simulation using the GUPFREQ keyword.

The GWRTWCV keyword should be followed by a data record for each gradient parameter. Each data record consists of one or more well names, and is terminated with a slash. The set of records terminates with a blank record containing just a slash (/).

Subsequent use of the GWRTWCV resets all the previous parameters.

Note: The current implementation of GWRTWCV is restricted to the standard well model and unrefined grids.

Example

The following example will set the first gradient parameter to be the control variable for the well PROD1, generating the gradient in the phase flow rates and well bottom hole pressure for all wells with respect to variations in the control variable of PROD1. This gradient data is available via the mnemonics WOFWC1, WWFWC1, WGFWC1 and WBHWC1. The second parameter will result in gradient data calculated with respect to simultaneously varying the control variables of both well PROD1 and well PROD2.

```plaintext
GWRTWCV
  PROD1 / -- parameter 1, output available via W**WC1 mnemonics
  PROD1 PROD2 / -- parameter 2
  INJ1 / -- parameter 3
/
-- later in deck redefine parameter 1 and reset parameters 2 & 3
GWRTWCV
  PROD3 /
/```

Keywords

1054
This section contains an alphabetic listing of keywords starting with the letter H.
Allow H2S to dissolve in the aqueous phase

This keyword turns on the H$_2$S solution option. The H$_2$S component is then allowed to exist in all three phases. This keyword cannot be used if directional saturation table end-point scaling is used (switched on by setting the first item of keyword ENDScale to DIRECT).

For further information see "CO$_2$/H$_2$S solution in the aqueous phase" in the ECLIPSE Technical Description.
HALFTRAN Specifies half block transmissibilities

The use of HALFTRAN implies that transmissibilities are calculated using half block permeabilities.

The half block permeabilities are not used directly in the simulator. The half block permeabilities and cell face transmissibility multipliers are assumed to have been processed into the MULTX and MULTX- arrays prior to exporting the data into ECLIPSE 100. This is implemented by overloading MULTX and MULTX- for example, to the ratio of half-block to block center permeabilities, which is performed in FloGrid.

See "Transmissibility calculations" in the ECLIPSE Technical Description for more information.

The HALFTRAN keyword has no associated data.
Additive modifiers for gradient parameters

These keywords are used to specify additive cumulative modifiers to history matching parameters. Their use in the ECLIPSE data file is embedded within the `HMPROPS` keyword, where examples of their usage may also be found.
**HDISP**

**Tracer mechanical dispersion data**

The **HDISP** keyword inputs the tracer mechanical dispersivity; the longitudinal value and the transverse value. The model assumes that the relative importance of diffusion and advection are independent of flow velocity and so can be characterized by a length scale, the mechanical dispersivity. The keyword should be concatenated with the tracer name. **HDISP** cannot be used in parallel runs and is only compatible with the standard and TVD tracer solvers.

For example: **HDISPXXX** is the keyword required to input the dispersivity for tracer **XXX** as specified in the **TRACER** keyword.

The data comprises **NTPVT** (see keyword **TABDIMS**) tables of diffusion data, each terminated by a slash (/). Each table applies to its corresponding PVT region, defined by keyword **PVTNUM** (if **NTPVT** > 1).

Each table consists of the following items of data:

1. The tracer mechanical dispersivity in the longitudinal (streamwise) direction.
   
   **UNITS:** m (METRIC), ft (FIELD), cm (LAB)

2. The tracer mechanical dispersivity in the transverse direction (perpendicular to the stream).
   
   **UNITS:** m (METRIC), ft (FIELD), cm (LAB)

**Note:** The use of **HDISP** can cause some runs to become much slower. In such cases it may be worth using the **SAMG** keyword.

**Example**

With **NTPVT**=2 for tracers WT1 and WT2

```
<table>
<thead>
<tr>
<th>HDISPWT1</th>
</tr>
</thead>
<tbody>
<tr>
<td>100 3 /</td>
</tr>
<tr>
<td>35 1 /</td>
</tr>
<tr>
<td>HDISPWT2</td>
</tr>
<tr>
<td>80 0 /</td>
</tr>
<tr>
<td>35 0 /</td>
</tr>
</tbody>
</table>
```
HEATCR

Rock heat capacity

This keyword can only be used with the **THERMAL** option.

**HEATCR** specifies the rock volumetric heat capacity value for each cell. Each keyword should be followed by \( N_x N_y N_z \) values in normal order, with the \( X \) index changing fastest.

**UNITS:** kJ/m\(^3\)/K (**METRIC**), Btu/ft\(^3\)/°R (**FIELD**), J/cc/K (**LAB**), kJ/m\(^3\)/K (**PVT-M**)

See "Thermal option" in the *ECLIPSE Technical Description* for further information on this option.

**Example**

Set heat capacities for a 10*10 cross section, with two different rock types.

```plaintext
HEATCR
--
   --In Field units
   60*35 40*30 /
```
HEATCRT Temperature dependence of the rock heat capacity

This keyword can only be used with the THERMAL option.

HEATCRT specifies the temperature dependence of the rock volumetric heat capacity value for each cell.

The enthalpy per unit volume of rock in each cell is calculated as

\[ H_{\text{rock}} = C_{R0}(T-T_{\text{ref}}) + \frac{C_{R1}(T-T_{\text{ref}})^2}{2} \]

where \( C_{R0} \) is given by the HEATCR keyword and \( C_{R1} \) is given by the HEATCRT keyword.

Each keyword should be followed by \( N_x N_y N_z \) values in normal order, with the X index changing fastest.

UNITS: kJ/m\(^3\)/K\(^2\) (METRIC), Btu/ft\(^3\)/°R\(^2\) (FIELD), J/cc/K\(^2\) (LAB), kJ/m\(^3\)/K\(^2\) (PVT-M)

See "Thermal option" in the ECLIPSE Technical Description for further information on this option.

Example

Set coefficients for a 10\(^*\)10 cross section, with two different rock types.

```
HEATCRT
60*0.0002   40*0.0001   /
```
HEATDIMS  Heater dimensions

This keyword can only be used with the THERMAL option (see "Thermal option" in the ECLIPSE Technical Description).

The data consists of the item, describing the dimensions of the heater data to be used in the run. The heater data for each connection should be defined using keyword HEATER or keyword HEATERL.

1. The maximum number of heater connections
   DEFAULT: 1

Example

HEATDIMS
10 /

HEATER

Heater data

This keyword can only be used with the THERMAL option.

HEATER specifies the position and properties of one or more heater connections. If the connection is in a local grid, then use keyword HEATERL.

The maximum number of heater connections must be set using keyword HEATDIMS.

This keyword is followed by any number of records, each containing the items of data described below, and each terminated by a / character.

The records can be terminated early using the / character, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n* where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. The name of the heater, a character string of up to 8 letters
2. The I-index of the connection
3. The J-index of the connection
4. The K-index of the connection
5. The maximum heat injection rate, \( H_{\text{max}} \)
   - UNITS: \( \text{kJ/day (METRIC)}, \text{Btu/day (FIELD)}, \text{J/hr (LAB)}, \text{kJ/day (PVT-M)} \)
   - DEFAULT: 0
6. The maximum temperature of the connection cell, \( T_{\text{max}} \)
   - UNITS: \( ^\circ\text{C (METRIC)}, ^\circ\text{F (FIELD)}, ^\circ\text{C (LAB)}, ^\circ\text{C (PVT-M)} \)
7. Temperature-dependent heat injection rate, \( R \)
   - UNITS: \( \text{kJ/day/K (METRIC)}, \text{Btu/day/°R (FIELD)}, \text{J/hr/K (LAB)}, \text{kJ/day/K (PVT-M)} \)

The heater can operate in up to three different modes depending upon the values assigned to items 5, 6 and 7:

Mode 1: Constant energy injection rate

If only item 5 is specified and items 6 and 7 are defaulted, heat will be injected at a constant rate into the grid block according to:

\[ H = H_{\text{max}} \]

where \( H_{\text{max}} \) is defined by item 5.

Mode 2: Energy density dependent injection rate

If items 5 and 6 are specified and item 7 is defaulted, heat will be injected at a rate determined by the product of the grid-block volume and the rate of change of energy density:

\[ H = \min\left(H_{\text{max}}, V \cdot \frac{\Delta E}{\Delta t}\right) \]

where \( V \) is the grid-block volume, \( \Delta t \) is the timestep, and the estimated change in energy density \( \Delta E \) is given by:

\[ \Delta E = e_t(T_{\text{max}}) - e_{t-\Delta t} \]
where \( e(T_{max}) \) is the grid-block energy density when its temperature is \( T_{max} \) and \( e_{t-\Delta t} \) is the grid-block energy density at the previous timestep. \( H_{max} \) and \( T_{max} \) are defined by items 5 and 6 respectively. If item 6 is set to a value greater than 1.0E+10, this will have the effect of defaulting item 6 such that the heat injection mechanism reverts to Mode 1, that is, a constant energy injection rate.

### Mode 3: Temperature difference dependent injection rate

If items 5, 6 and 7 are all defined, heat will be injected at a rate determined by the temperature difference between the maximum temperature of the connection \( T_{max} \) and the current grid-block temperature \( T_c \).

\[
H = \min\left( (T_{max} - T_c) \times R, H_{max} \right)
\]

where \( H_{max}, T_{max} \) and \( R \) are defined by items 5, 6 and 7 respectively.

**Note:** If item 5 is defaulted, the heater will be turned off irrespective of whether items 6 and 7 are specified or defaulted. If item 5 is specified and item 6 is defaulted, the heater will be deemed to be operating in Mode 1 irrespective of whether item 7 is specified or defaulted.

**Note:** If items 5, 6 and 7 are defined with item 7 set to zero, then \( H = \min\left[0, H_{max}\right] \). If \( H_{max} \) (item 5) is greater than 0, then the heater will be deemed to be operating in Mode 1 and turned off.

The output of heater performance data to the print file can be controlled by using item 14 of the `RPTPRINT` keyword.

Summary vectors `WHTR` and `WHTT` contain the energy rate and energy total for individual heaters and require a list of heater names. Vectors `FHTR` and `FHTT` contain the energy rate and energy total for all heaters.

**Example**

```plaintext
HEATER
HEAT1 16  8 2 3.E+6 500.0 /
HEAT1 16  8 3 5.E+6 500.0 /
HEAT2 22 45 5 1.E+6 1* /
```

**Keywords**

1064
HEATERL LGR heater data

This keyword can only be used with the THERMAL option.

HEATERL specifies the position and properties of one or more heater connections in a local grid refinement. If the connection is not in a local grid, then use keyword HEATER.

The maximum number of heater connections must be set using keyword HEATDIMS.

This keyword is followed by any number of records, each containing the items of data described below, and each terminated by a / character.

The records can be terminated early using the / character, the remaining items taking default values.

Defaults can also be specified by a null repeat count of the form n* where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. The name of the heater, a character string of up to 8 letters
2. The name of the local grid
3. The I-index of the connection
4. The J-index of the connection
5. The K-index of the connection
6. The maximum heat injection rate, $H_{max}$
   - UNITS: kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB), kJ/day (PVT-M)
   - DEFAULT: 0
7. The maximum temperature of the connection cell, $T_{max}$
   - UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
8. Temperature-dependent heat injection rate, $R$
   - UNITS: kJ/day/K (METRIC), Btu/day/°R (FIELD), J/hr/K (LAB), kJ/day/K (PVT-M)

The heater can operate in up to three different modes depending upon the values assigned to items 6, 7 and 8:

**Mode 1: Constant energy injection rate**

If only item 6 is specified and items 7 and 8 are defaulted, heat will be injected at a constant rate into the grid block according to:

$$H = H_{max}$$

where $H_{max}$ is defined by item 6.

**Mode 2: Energy density dependent injection rate**

If items 6 and 7 are specified and item 8 is defaulted, heat will be injected at a rate determined by the product of the grid-block volume and the rate of change of energy density:

$$H = \min \left( H_{max}, V \cdot \frac{\Delta E}{\Delta t} \right)$$
where $V$ is the grid-block volume, $\Delta t$ is the timestep, and the estimated change in energy density $\Delta E$ is given by:

$$\Delta E = e_t(T_{\text{max}}) - e_{t-\Delta t}$$

where $e_t(T_{\text{max}})$ is the grid-block energy density when its temperature is $T_{\text{max}}$ and $e_{t-\Delta t}$ is the grid-block energy density at the previous timestep. $H_{\text{max}}$ and $T_{\text{max}}$ are defined by items 6 and 7 respectively. If item 7 is set to a value greater than 1.0E+10, this will have the effect of defaulting item 7 such that the heat injection mechanism reverts to Mode 1, that is, a constant energy injection rate.

**Mode 3: Temperature difference dependent injection rate**

If items 6, 7 and 8 are all defined, heat will be injected at a rate determined by the temperature difference between the maximum temperature of the connection $T_{\text{max}}$ and the current grid-block temperature $T_c$.

$$H = \min\left([T_{\text{max}}-T_c] \times R, \ H_{\text{max}}\right)$$

where $H_{\text{max}}$, $T_{\text{max}}$ and $R$ are defined by items 6, 7 and 8 respectively.

**Note:** If item 6 is defaulted, the heater will be turned off irrespective of whether items 7 and 8 are specified or defaulted. If item 6 is specified and item 7 is defaulted, the heater will be deemed to be operating in Mode 1 irrespective of whether item 8 is specified or defaulted.

**Note:** If items 6, 7 and 8 are defined with item 8 set to zero, then $H = \min\left([0, \ H_{\text{max}}]\right)$ If $H_{\text{max}}$ (item 6) is greater than 0, then the heater will be deemed to be operating in Mode 1 and turned off.

The **RPTPRINT** keyword can be used to output heater status to the Print file.

**Example**

```
HEATERL
Heat1 LGR1 6 9 4 3.E+6 1* /
```

---

**Keywords**

1066
HEATTR

Heat transmissibilities for radial run

This keyword can only be used with the THERMAL option.

HEATTR specifies the thermal transmissibility values in the radial direction, replacing those calculated by the program.

Each keyword should be followed by $N_rN_\thetaN_z$ values in natural order with the R-index changing fastest. In each case, the transmissibility is that between a cell and its neighbor in the positive direction.

UNITS: kJ/day/K (METRIC), Btu/day/°R (FIELD), J/hr/K (LAB), kJ/day/K (PVT-M)

Example

HEATTR
--
--In Field units
60*35 40*30 /
HEATTHT  Heat transmissibilities for radial run

This keyword can only be used with the THERMAL option.

HEATTHT specifies the thermal transmissibility values in the azimuthal direction, for runs using radial geometry.

Each keyword should be followed by $N_r, N_\theta, N_z$ values in natural order.

UNITS: kJ/day/K (METRIC), Btu/day/°R (FIELD), J/hr/K (LAB), kJ/day/K (PVT-M)
HEATTX

Heat transmissibility x-direction

This keyword can only be used with the THERMAL option.

HEATTX specifies the thermal transmissibility values in the X direction, replacing those calculated by the program.

The heat transmissibility is the conductive heat transfer transmissibility and is obtained by multiplying the thermal conductivity of the rock by the cross sectional area divided by the block center to block center distance.

The thermal transmissibilities are entered as a number for each grid cell - that is $N_x \times N_y \times N_z$ in all, in normal grid order with the X-index changing fastest. The thermal transmissibility is that between a cell and its neighbor in the positive direction.

UNITS: $kJ/\text{day/K}$ (METRIC), $\text{Btu/day/°R}$ (FIELD), $J/\text{hr/K}$ (LAB), $kJ/\text{day/K}$ (PVT-M)

Example

```
HEATTX
--In Field units
60*35 40*30 /
```
HEATTY

Heat transmissibility y-direction

This keyword can only be used with the THERMAL option.

HEATTY specifies the thermal transmissibility values in the Y direction, replacing those calculated by the program.

The thermal transmissibilities are entered as a number for each grid cell - that is $N_x N_y N_z$ in all, in normal grid order with the X-index changing fastest. The thermal transmissibility is that between a cell and its neighbor in the positive direction.

UNITS: kJ/day/K (METRIC), Btu/day/°R (FIELD), J/hr/K (LAB), kJ/day/K (PVT-M)
HEATTZ

Heat transmissibility z-direction

This keyword can only be used with the THERMAL option.

HEATTZ specifies the thermal transmissibility values in the Z direction, replacing those calculated by the program.

The thermal transmissibilities are entered as a number for each grid cell - that is \( N_x N_y N_z \) in all, in normal grid order with the X-index changing fastest. The thermal transmissibility is that between a cell and its neighbor in the positive direction.

UNITS: kJ/day/K (METRIC), Btu/day/°R (FIELD), J/hr/K (LAB), kJ/day/K (PVT-M)
Heat of vaporization

This keyword can only be used with the THERMAL option.

HEATVAP specifies the constant part of the heat of vaporization of each oil component. This keyword should only be used for thermal runs with volatile oils; it should not be used with DEADOIL.

The data record contains one value for each component:

**UNITS:** kJ/kg (METRIC), Btu/lb (FIELD), J/gm (LAB), kJ/kg (PVT-M)

**DEFAULT:** 0.0

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

The enthalpy of a gaseous oil component may be calculated by first determining the enthalpy of the component in its liquid state, then adding a heat of vaporization of the form

\[ \Delta H = A \cdot \left(1 - \frac{T}{T_{crit}}\right)^B \]

where

- \( A \) Constant part of \( \Delta H \) entered with this keyword
- \( B \) Exponent entered with keyword HEATVAPE (normally default)
- \( T \) Temperature
- \( T_{crit} \) Critical temperature of the component,keyword TCRIT.

Heats of vaporization are often obtained at the normal boiling point. Hence the constant \( A \) above would be entered as

\[ A = \frac{\Delta H_{nb}}{(1-T_{nb}/T_{crit})^B} \]

where \( \Delta H_{nb} \) is the heat of vaporization at the normal boiling point, and \( T_{nb} \) is the normal boiling point. The exponent \( B \), entered with keyword HEATVAPE, is usually defaulted to 0.38.

**Note:** Only two of the three keywords SPECHG, SPECHA and HEATVAP should be specified. For further information please see "Enthalpy" in the ECLIPSE Technical Description.

Example

For 4 components of molecular weight 16, 134, 394, 539

 FIELD
 HEATVAP
 355 185 98 73/

Keywords

1072
HEATVAPE  Heat of vaporization exponent

This keyword can only be used with the THERMAL option.

HEATVAPE specifies the exponent to be used in the heat of vaporization calculation. This keyword should only be used for thermal runs with volatile oils; it should not be used with DEADOIL.

The data record contains one value for each component:

DEFAULT: 0.38

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as the 9th parameter of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state" in the ECLIPSE Technical Description.

The enthalpy of a gaseous oil component may be calculated by first determining the enthalpy of the component in its liquid state, then adding a heat of vaporization of the form

$$ \Delta H = A \cdot (1 - T / T_{crit})^B $$

Eq. 3.82

where

- $A$ Constant part of $\Delta H$ entered with keyword HEATVAP
- $B$ Exponent entered with this keyword (normally default)
- $T$ Temperature
- $T_{crit}$ Critical temperature of the component, keyword TCRIT.

Example

For three components:

```
HEATVAPE 1.0 1.0 1*
```

ECLIPSE Reference Manual

Keywords

1073
HEATVAPS  Heat of vaporization at standard temperature

This keyword can only be used with the THERMAL option.

HEATVAPS specifies the heat of vaporization of each component at the standard temperature (STCOND). This keyword should only be used for thermal runs with volatile oils; it should not be used with DEADOIL. The keyword can be used when both SPECHA and SPECHG keywords are used. It should not be used if the HEATVAP keyword is used.

The data record contains one value for each component:

UNITs: kJ/kg (METRIC), Btu/lb (FIELD), J/gm (LAB), kJ/kg (PVT-M)

DEFAULT: 0.0

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of TABDIMS. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

Example

For 2 components

| HEATVAPS | 1000.0 2000.0/ |
**Specify gradients for Carter-Tracy aquifers**

The keyword can be used to obtain gradients of the solution with respect to parameters defining the performance of a Carter-Tracy aquifer. The analytic aquifer should previously have been defined using keywords `AQUCT` and `AQUANCON` or `AQANCONL` in the `SOLUTION` section.

The keyword should be followed by a number of records defining the aquifers and gradient parameters for each aquifer.

The number of derivative calculations for analytic aquifers must not exceed `MHAQUF` specified in keyword `HMDIMS`.

The set of records must end with a single line containing only a slash (/).

Each record refers to a separate Carter-Tracy aquifer, and contains the following items of data:

1. **Aquifer identification number**
   - The identification number is specified as item 1 `AQUCT`.

2. **Calculate derivatives with respect to a permeability multiplier for the aquifer:**
   - `YES` derivatives with respect to a multiplier on the aquifer permeability
   - `NO` do not evaluate derivatives with respect to a multiplier on the aquifer permeability
   - **DEFAULT:** `NO`

3. **Calculate derivatives with respect to a multiplier on the aquifer opening angle**
   - `YES` derivatives with respect to a multiplier on the aquifer opening angle are evaluated:
   - `NO` do not evaluate derivatives with respect to a multiplier on the aquifer opening angle
   - **DEFAULT:** `NO`

4. **Calculate derivatives with respect to the aquifer depth:**
   - `YES` derivatives with respect to the aquifer depth are evaluated
   - `NO` do not evaluate derivatives with respect to the aquifer depth
   - **DEFAULT:** `NO`

If the initial aquifer pressure is not defaulted in keyword `AQUCT` then the initial derivative of aquifer pressure with respect to depth is set to zero; if the aquifer pressure is defaulted, the initial derivative is calculated from the assumption that the aquifer is in equilibrium with the reservoir. In this case the gradients with respect to aquifer depth will be insensitive and hence redundant in a regression.

In the absence of brine, the above parameters constitute a set of independent parameters defining the aquifer inflow performance. See "Aquifer modeling facilities" in the *ECLIPSE Technical Description*.

**Example**

With `MHAQUF≥2` specified using `HMDIMS`
<table>
<thead>
<tr>
<th>no. depth</th>
<th>pressure</th>
<th>knots of inf. table</th>
<th>conc.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>7000</td>
<td>4000</td>
<td>20.0</td>
</tr>
</tbody>
</table>

Aquifer Derivative w.r.t. Permeability multiplier opening angle multiplier

---

---

HMAQUCT

ECLIPSE Reference Manual

Keywords

1076
HMAQUFET  Specify gradients for Fetkovich aquifers

The keyword can be used to obtain gradients of the solution with respect to parameters defining the performance of a Fetkovich aquifer. The analytic aquifer should previously have been defined using either keyword AQUFET or both keywords AQUFETP and AQUANCON or AQAANCONL in the SOLUTION section.

The keyword should be followed by a number of records defining the aquifers and gradient parameters for each aquifer.

The number of derivative calculations for analytic aquifers must not exceed MHAQUF specified in keyword HMDIMS.

Each record is terminated by a slash (/). The set of records must be terminated with a blank record containing just a slash.

Each record refers to a separate Fetkovich aquifer, and contains the following items of data:

1. Aquifer identification number
   - If the AQUFETP/AQUANCON or AQUFETP/AQANCONL combination is used to define Fetkovich aquifers, the identification number is specified in AQUFETP item 1. Otherwise, the aquifer is identified by its record number in AQUFET.

2. Calculate derivatives with respect to a multiplier on the initial volume of water in the aquifer
   - YES derivatives with respect to a multiplier on the initial aquifer volume are evaluated
   - NO do not evaluate derivatives with respect to multiplier on the initial aquifer volume
   - DEFAULT: NO

3. Calculate derivatives with respect to aquifer productivity index multiplier
   - YES derivatives with respect to the productivity index multiplier are evaluated
   - NO do not evaluate derivatives with respect to the productivity index multiplier
   - DEFAULT: NO

4. Calculate derivatives with respect to the aquifer depth
   - YES derivatives with respect to the aquifer depth are evaluated
   - NO do not evaluate derivatives with respect to the aquifer depth
   - DEFAULT: NO

If the initial aquifer pressure is not defaulted in keyword AQUFET then the initial derivative of aquifer pressure with respect to depth is set to zero; if the aquifer pressure is defaulted, the initial derivative is calculated from the assumption that the aquifer is in equilibrium with the reservoir. In this case the gradients with respect to aquifer depth will be insensitive and hence redundant in a regression.

In the absence of brine, the above parameters constitute a set of independent parameters defining the aquifer inflow performance. See “Aquifer modeling facilities” in the ECLIPSE Technical Description.

The keyword must not be used in restarted runs, as the aquifer specification data cannot be changed.
Examples

Example 1

AQUCT
-- Aq. Datum Initial Perm Poro Ct Ro Thick-Angle Water AQUTAB Salt
-- no. depth pressure ness of inf. table table conc.
  1 7000 4000 200.0 0.2 1.0E-5 500.0 50.0 20.0 1 1 0.0 /

HMAQUFET
-- Aquifer Derivative w.r.t. Derivative w.r.t.
-- number volume multiplier P.I. multiplier
  1 YES YES /

Example 2

One history matching aquifer, on the bottom of a 10 x 10 x 5 grid with $\text{MHAQUF} \geq 2$ specified in keyword HMDIMS.

AQUFET
-- Datum Initial Wi Ct J Table I1 I2 J1 J2 K1 K2 Face
-- depth pressure for PVTW
  7000 4000 2.0E9 1.0E-5 500.0 1 1 10 1 10 5 5 'K+' /

HMAQUFET
-- Aquifer Derivative w.r.t. Derivative w.r.t.
-- number volume multiplier P.I. multiplier
  1 YES YES /

HMAQUNUM  Calculate gradients for numerical aquifers

The keyword can be used to obtain gradients of the solution with respect to parameters defining the performance of a numerical aquifer. The aquifer should previously have been defined using keywords AQUNUM and AQUCON in the GRID section.

The keyword should be followed by a number of records defining the aquifers and gradient parameters for each aquifer.

The number of derivative calculations defined for all analytic and numerical aquifers must not exceed MHAQUF specified in item 5 of keyword HMDIMS.

The set of records must end with a single line containing only a slash (/).

Each record refers to a separate numerical aquifer, and contains the following items of data (and is followed by a slash):

1. Aquifer identification number
   The identification number is specified as item 1 in keyword AQNUM.

2. Calculate derivatives with respect to a pore volume multiplier for the aquifer
   YES  Derivatives with respect to a multiplier on the aquifer pore volume are evaluated
   NO   Do not evaluate derivatives with respect to a multiplier on the aquifer pore volume
   DEFAULT: NO

3. Calculate derivatives with respect to a multiplier on the aquifer permeability
   YES  Derivatives with respect to a multiplier on the aquifer permeability are evaluated
   NO   Do not evaluate derivatives with respect to a multiplier on the aquifer permeability

4. Calculate derivatives with respect to the aquifer-grid connection transmissibility
   YES  Derivatives with respect to the aquifer transmissibility are evaluated
   NO   Do not evaluate derivatives with respect to the aquifer transmissibility
   DEFAULT: NO

Note: The pore volume parameter acts over all cells comprising a single aquifer.
The permeability parameter relates to a multiplier on the permeability of all cells representing the aquifer. The transmissibility parameter relates to a multiplier on the transmissibilities only of the aquifer-grid connections and not for the aquifer-aquifer connections when an aquifer is represented by several cells. This acts like the transmissibility multiplier specified with item 9 in the AQUCON keyword.

Note: The pore volume of a numerical aquifer cell is calculated from \( \text{PORV} = \text{Length} \times \text{Area} \times \text{Por} \) using data entered in the AQNUM keyword, where the porosity Por is taken from the grid block value if defaulted. The transmissibilities of numerical aquifer connections are calculated from the permeability entered in the AQNUM keyword. The regional parameter HMPORVM may include aquifer cells to obtain sensitivities, but any values of HMULTPV will be ignored for numerical aquifer cells during a regression.
from SimOpt because the aquifer PORV always uses the above formula and ignores pore volume modifiers. Keyword HMMLAQUN should be used to modify the aquifer pore volume during a regression run.

The regional parameters, for example HMPERMX, return zero sensitivities for any aquifer cells because the aquifer permeability is taken directly from AQUNUM and not from the grid values or permeability. ECLIPSE will remove all numerical aquifer cells from regions defined by HMPORVM, HMPERMX and related keywords, because HMAQUNUM should be used instead.

Sensitivities for transmissibilities between different numerical aquifers or for groups of numerical aquifer to grid connections can be obtained using the inter-region transmissibility parameter HMULRGT by defining MULTNUM for the aquifer cells specified in AQUNUM.

For further information on the treatment of numerical aquifers, see "Aquifer modeling facilities" in the ECLIPSE Technical Description.

Example

With MHAQUF>=1 specified in HMDIMS.

<table>
<thead>
<tr>
<th>AQUNUM</th>
<th>-- Aq. IX IY IZ Area</th>
<th>Length</th>
<th>Poro.</th>
<th>Perm.</th>
<th>Depth</th>
<th>Pressure</th>
<th>PVT</th>
<th>Sat.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>7 1 1 1.0E+5 2000</td>
<td>1*</td>
<td>200.0</td>
<td>1*</td>
<td>1*</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>1</td>
<td>8 1 1 1.0E+5 2000</td>
<td>1*</td>
<td>200.0</td>
<td>1*</td>
<td>1*</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>1</td>
<td>9 1 1 1.0E+5 2000</td>
<td>1*</td>
<td>200.0</td>
<td>1*</td>
<td>1*</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>1</td>
<td>10 1 1 1.0E+5 2000</td>
<td>1*</td>
<td>200.0</td>
<td>1*</td>
<td>1*</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>HMAQUNUM</th>
<th>-- Aquifer</th>
<th>PoreV parameter</th>
<th>Perm. parameter</th>
<th>Trans. parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>YES</td>
<td>YES</td>
<td>NO</td>
<td>/</td>
</tr>
</tbody>
</table>

Keywords
**HMDIMS**

Specify the dimensions for the gradient option

This keyword specifies the dimensions for the gradient option, and is followed by some or all of the following items, terminated by a slash (/)

1. The maximum number of gradient regions defined by keywords HMxxxxxx in the REGIONS section
   DEFAULT: 0

2. The maximum subregion number occurring in any Gradient region
   DEFAULT: 0

3. The maximum number of gradients required for Gradient region keywords.
   DEFAULT: If this is not specified, it defaults to MHISTM * NMHSTR, where MHISTM is the maximum number of Gradient regions specified by item 1 of this keyword and NMHSTR the maximum subregion number occurring in any Gradient region specified by item 2 of this keyword.

   When NMHSTR is large, the memory required can therefore be reduced by specifying NHRPAR, which is actual maximum number of active ‘gradzone’ parameters.

4. The maximum number of faults for which gradients are required, specified by keyword HMFAULTS in the GRID section, and also the maximum number of inter-region connection gradients specified using keyword HMMULRGT in the GRID section.
   DEFAULT: 0

5. The maximum number of parameters declared for aquifers for which gradients are required specified by keywords HMAQUCT or HMAQUFET in the SOLUTION section or by keyword HMAQUNUM in the GRID section.
   DEFAULT: 0

6. The maximum number of well parameters specified using keyword HMWPIMLT in the SCHEDULE section.
   DEFAULT: 0

7. Not used at present

8. The maximum number of parameters declared for gradients with respect to rock parameters (keywords HMROCK and HMROCKT in the PROPS section).
   DEFAULT: 0

9. The maximum number of parameters declared for gradients with respect to well connection parameters (that is, using the keyword HMWELCON in the SOLUTION section).
   DEFAULT: 0

**Note:** ECLIPSE requires additional memory, which is a minimum of approximately 3 double precision words per cell for each gradient parameter. Hence, to make best of the available memory, it is not advisable to set the maximum number of gradient parameters greater than the actual number to be used in each case.

See "Gradient option" in the *ECLIPSE Technical Description* for further information.
Examples

Example 1

In the following ECLIPSE 100 example there is a maximum of 3 gradient regions, each with up to 4 subregions with only 6 gradients required and 5 faults (with gradients required for two faults). There are two analytic aquifers with up to 100 grid block connections each and a default influence table.

```
FAULTDIM
---MFSEGS
  5 /
AQUDIMS
---MXNAQN MXNAQC NIFTBL NRIPTB NANAQU NCAMAX
  0  0  1  36  2  100 /
HMDIMS
---MHISTM NHMSTR NHRPAR MFHIST MHAQUF
  3  4  6  2  2 /
```

Example 2

In the following example there is a maximum of 4 gradient regions, each with up to 5 subregions with only 8 gradients required.

```
HMDIMS
---MHISTM NHMSTR NHRPAR
  8 /
```
**HMFAULTS  Specify gradients for faults**

This keyword may be used in runs in which the gradient option is required.

The keyword can be used to obtain gradients of the solution with respect to transmissibilities across faults which have been previously defined by the FAULTS keyword in the GRID section.

The keyword should be followed by a number of records defining names of fault segments for which solution gradients are required.

The set of records must end with a single line containing only a slash (/). The set of records must be terminated with a blank record containing just a slash.

The number of gradients for faults must not exceed MFHIST specified in RUNSPEC keyword HMDIMS.

**Example**

```
HMFAULTS
'zigzag' /
'block' /
```

**Keywords**

<table>
<thead>
<tr>
<th>x</th>
<th>ECLIPSE 100</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>ECLIPSE 300</td>
</tr>
<tr>
<td>x</td>
<td>SPECIAL</td>
</tr>
<tr>
<td></td>
<td>RUNSPEC</td>
</tr>
<tr>
<td>x</td>
<td>GRID</td>
</tr>
<tr>
<td></td>
<td>EDIT</td>
</tr>
<tr>
<td></td>
<td>PROPS</td>
</tr>
<tr>
<td></td>
<td>REGIONS</td>
</tr>
<tr>
<td></td>
<td>SOLUTION</td>
</tr>
<tr>
<td></td>
<td>SUMMARY</td>
</tr>
<tr>
<td></td>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>
HMMLAQUN  Multipliers for numerical aquifer properties

This keyword applies multipliers to properties determining the performance of a numerical aquifer defined in keyword AQNUM. It is intended to be used as a modifier on the base data specified in the AQNUM keyword during history matching.

The keyword should be followed by a number of records defining the aquifers and multipliers for each aquifer.

The set of records must end with a single line containing only a slash (/). The set of records must be terminated with a blank record containing just a slash.

Each record refers to a separate numerical aquifer, and contains the following items of data:

1. Aquifer identification number
   The identification number is specified in AQNUM item 1.

2. Multiplier on aquifer pore volume
   DEFAULT: 1.0

3. Multiplier on aquifer permeability
   DEFAULT: 1.0

   Note: The aquifer-grid connection transmissibility is also modified when the aquifer permeability multiplier is used. For further information on the transmissibility calculation see "Aquifer modeling facilities" in the ECLIPSE Technical Description.

4. Multiplier on aquifer-grid connection transmissibility
   DEFAULT: 1.0

See also the HMAQNUM keyword.

Note: The aquifer pore volume cannot be modified by editing PORV for the aquifer cells or by specifying MULTPV. See the AQNUM keyword.

Example

```
AQNUM
  -- Aq. IX IY IZ Area  Length Poro. Perm. Depth Pressure PVT Sat.
  1  7  1  1 1.0E+5  2000  1*  200.0  1*  1*  1  1 /
  1  8  1  1 1.0E+5  2000  1*  200.0  1*  1*  1  1 /
  1  9  1  1 1.0E+5  2000  1*  200.0  1*  1*  1  1 /
  1 10  1  1 1.0E+5  2000  1*  200.0  1*  1*  1  1 /
/
HMMLAQUN
  -- Aquifer PoreV parameter Perm. parameter Trans. parameter
     1  1.01  1.3  1.0 /
/
```

Keywords
Specify modifiers for Carter-Tracy aquifers

This keyword applies multipliers to properties determining the performance of the Carter-Tracy aquifer defined in keyword \textit{AQUCT}. It is intended to be used as a modifier on the base data specified in the \textit{AQUCT} keyword during history matching.

The keyword should be followed by a number of records defining the aquifers and multipliers for each aquifer.

The set of records must end with a single line containing only a slash (/). The set of records must be terminated with a blank record containing just a slash.

Each record refers to a separate Carter-Tracy aquifer, and contains the following items of data:

1. Aquifer identification number
   
   The identification number is specified in \textit{AQUCT} item 1.

2. Multiplier on aquifer permeability.
   
   DEFAULT: 1.0

   
   DEFAULT: 1.0

4. Multiplier on aquifer depth
   
   DEFAULT: 1.0

See also the \textit{HMAQUCT} keyword.

Example

\begin{verbatim}
AQUCT
---Aq. Datum Initial Perm Poro Ct Ro Thick- Angle Water AQUTAB Salt
---no. depth pressure ness of inf. table table conc.
1 7000 4000 200.0 0.2 1.0E-5 500.0 50.0 20.0 1 1 0.0 /
/
HMMLCTAQ
-- Aquifer Permeability Influence angle
-- number multiplier multiplier
1 2.0 1.5 /
/
\end{verbatim}
**Specify modifiers for Fetkovich aquifers**

This keyword applies multipliers to properties determining the performance of the Fetkovich aquifer defined using either keyword `AQUFET`, or keywords `AQUFETP` and `AQUANCON` or `AQANCONL`. It is intended to be used as a modifier on the base data specified in the `AQUFET` keyword during history matching.

The keyword should be followed by a number of records defining the aquifers and multipliers for each aquifer.

The set of records must end with a single line containing only a slash (/). The set of records must be terminated with a blank record containing just a slash.

Each record refers to a separate Fetkovich aquifer, and contains the following items of data:

1. Aquifer identification number. The identification number is specified in `AQUFETP` item 1 or is the number in the list if keyword `AQUFET` is used.
2. Multiplier on initial volume of water in the aquifer.
   - DEFAULT: 1.0
3. Multiplier on aquifer Productivity Index.
   - DEFAULT: 1.0
4. Multiplier on aquifer depth
   - DEFAULT: 1.0

See also the `HMAQUFET` keyword.

**Example**

Obtain gradients for one aquifer, on the bottom of a 10 x 10 x 5 grid

```plaintext
AQUFET
-- Datum Initial Wi Ct J Table I1 I2 J1 J2 K1 K2 Face
-- depth pressure   for PVTW
 7000  4000  2.0E9  1.0E-5  500.0  1  1 10 1 10 5 5 'K+' /
/
HMMLFTAQ
-- Aquifer Initial volume P.I.
-- number multiplier multiplier
 1  2.0  3.0 /
/```

**Keywords**

- `AQUFET`
- `AQUFETP`
- `AQUANCON`
- `AQANCONL`
HMMLTWCN  Modifies well connection transmissibility factor and skin factor

This keyword can be used in runs in which the gradient option is required.

The keyword can be used to modify the connection transmissibility factor and skin factor for well connections that are defined using the COMPDAT (or COMPDATL or COMPDATM) keyword in the SCHEDULE section.

The multipliers specified are applied cumulatively, so if a well connection is specified more than once, or in more than one HMMLTWCN keyword, the multipliers will be cumulated.

The keyword is followed by any number of records, described below, each record terminated by a slash (/). A record can be terminated early using a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Well name
   This item cannot be defaulted. It should be the name of a single well which is defined in the SCHEDULE section. You cannot use a wildcard character to refer to more than one well at a time, and you cannot use a well list (that is, as defined using the keyword WLIST).

2. Grid name
   If the well connection is not in a local grid, then you should specify ‘FIELD’ for this item. If the well connection is in a local grid, this item should specify the name of the local grid refinement, and it should correspond to the local grid specified for the well connection in the SCHEDULE section.

   DEFAULT: ‘FIELD’

3. I - location of the connecting grid block
   This specifies the I-location of the well connection on the grid specified in item 2. It should correspond to the I-location of the connection specified in the SCHEDULE section.

   This item cannot be defaulted.

4. J - location of the connecting grid block
   This specifies the J-location of the well connection on the grid specified in item 2. It should correspond to the J-location of the connection specified in the SCHEDULE section.

   This item cannot be defaulted.

5. K - location of the connecting grid block
   This specifies the K-location of the well connection on the grid specified in item 2. It should correspond to the K-location of the connection specified in the SCHEDULE section.

   This item cannot be defaulted.

6. Modifier for the connection transmissibility factor.
   The connection transmissibility factor for the well connection will be multiplied by this value. It should be set to a value greater than zero.

   DEFAULT: 1.0
7. Modifier for the skin factor.

The skin factor for the well connection will be multiplied by this value. It should be set to a value greater than zero.

DEFAULT: 1.0

Example

This example assumes that the local grid, wells and connections are defined in the SCHEDULE section.

```
HMOMLTWCN
-- Well Grid I J K CTF Skin
PROD1 CARF1 6 4 1 1.01 0.95 /
PROD1 CARF1 6 4 2 1*  1.40 /
PROD1 CARF1 6 4 3 1.50 /
PROD3  1* 4 5 2 1.00 1.00 /
PROD3  1* 4 5 3 2.50 1* /
```
HMMLTxxx  Cumulative multipliers for gradient parameters

See keyword HMMLTxx.
HMMDREGT  Cumulative inter-region transmissibility modifiers

This is analogous to MULTREGT except the effect is cumulative.

The keyword is followed by any number of records, each terminated with a slash (/).

Each record contains the following items of data:

1. From region number (I)
2. To region number (J)
3. Additional transmissibility multiplier (TMLT) for all connections between FLUXNUM or MULTNUM regions I and J.

The set of records must end with a single line containing only a slash (/). The set of records must be terminated with a blank record containing just a slash.

The options specified in MULTREGT items 4 and 5 apply. If the regions I and J have not previously been declared using MULTREGT, then TMLT is applied to all connections between I and J (that is, items 4 and 5 of MULTREGT are assumed defaulted).

Example

```
HMMDREGT
  1  2  1.1 /
  3  4  1.5 /
/
```

Keywords

1090
**HMMROCK  Specify cumulative multipliers on rock compressibility**

The keyword can be used to specify modifiers to the rock compressibility entered via keyword ROCK, in each PVTNUM or SATNUM region.

The keyword should be followed by up to NTPVT (see item 2 of keyword TABDIMS) records defining the gradient calculations for each PVT region, or by up to NTSFUN (see item 1 in keyword TABDIMS) records if the ROCK keyword refers to saturation regions as requested in the ROCKOPTS keyword.

The set of records must end with a single line containing only a slash (/). The set of records must be terminated with a blank record containing just a slash.

Each record consists of two items:

1. PVT or saturation table number.
   - This should be in the range 1 to NTPVT (or in the range 1 to NTSFUN if rock data is entered by saturation region).

2. The compressibility multiplier \( \lambda \) for this table.
   - DEFAULT: 1.0

The pore volume is calculated as

\[
P V = P V_{ref}(1 + X + X^2)
\]

with

\[
X = C(p - p_{ref})
\]

and where \( C \) and \( p_{ref} \) are the rock compressibility and reference pressure entered in the ROCK keyword.

ECLIPSE calculates derivatives with respect to \( \lambda \), a multiplier on \( C \), so that

\[
X = \lambda C(p - p_{ref})
\]

**Note:** By default, rock compressibility data is entered by PVTNUM region. But if item 3 in keyword ROCKOPTS is set to SATNUM, the rock compressibility data is entered according to SATNUM region.

**Example**

```
HMMROCK
  2 1.03 /
  4 1.04 /
/
```
HMMROCKT  Specify cumulative modifiers for rock compaction parameters

The keyword can be used to specify modifiers to rock compaction parameters entered via keyword ROCKTAB or ROCKTABH in each ROCKNUM region.

The pore volume is calculated as a multiplier on the reference pore volume using data supplied in the ROCKTAB or ROCKTABH tables:

\[ P_V = P_{V_{\text{ref}}} M(p) \]

where \( p \) is the reservoir pressure. If the OVERBURD keyword has been entered,

\[ p = p_{\text{eff}} = p_{\text{fluid}} - P_{\text{overburden}} \]

If the STRESS option is selected in item 1 of the ROCKOPTS keyword,

\[ p = p_{\text{eff}} = P_{\text{overburden}} - p_{\text{fluid}} \]

ECLIPSE calculates gradients with respect to two parameters using the functional form

\[ P_V = P_{V_{\text{ref}}} M(p) \left( \lambda_1 + \lambda_2 \frac{(p - p_{\text{ref}})}{p_{\text{dim}}} \right) \]

where \( p_{\text{ref}} \) is the reference pivot pressure entered in keyword HMRREF for the ROCKNUM region and \( p_{\text{dim}} \) is a reference scale pressure used to make the parameter \( \lambda_2 \) dimensionless.

Derivatives are calculated with respect to the dimensionless parameters \( \lambda_1 \) and \( \lambda_2 \).

The keyword should be followed by up to NTROCC (item 2 of keyword ROCKCOMP) records defining the gradient calculations for each ROCKNUM region. Each record consists of three items and is followed by a slash(/). The set of records should be terminated by a single slash.

1. Rock table number
   This should be in the range 1 to NTROCC (item 2 of keyword ROCKCOMP).
2. Modifier \( \lambda_1 \) (multiplicative parameter)
   DEFAULT: 1.0
3. Modifier \( \lambda_2 \) (additive parameter)
   DEFAULT: 0.0

Example

```
HMMROCKT
  2 1.0 1.13 /
  4 1.0 1.05 /
```

Keywords

1092
**HMMULRGT**  
**Calculate gradients for inter-region transmissibility multipliers**

The keyword can be used to obtain gradients of the solution with respect to inter-region transmissibility multipliers declared using keyword **MULTREGT** in the GRID section.

The keyword is followed by any number of records, each terminated with a slash (/). The set of records must be terminated with a blank record containing only a slash.

Each record contains the following items of data:

1. From region number (I)
   - A non-positive or defaulted value acts as a wildcard on all regions.
   - DEFAULT: Negative

2. To region number (J)
   - A non-positive or defaulted value acts as a wildcard on all regions.
   - DEFAULT: Negative

3. Direction(s) in which to apply the multiplier:
   - Either X, Y, Z, XY, YZ, XZ or XYZ.
   - DEFAULT: XYZ

4. Flag governing application of the multiplier to non-neighbor connections:
   - NNC: Apply the transmissibility multiplier only to non-neighbor connections between regions I and J
   - NONNC: Do not apply the transmissibility multiplier to any non-neighbor connections between regions I and J
   - ALL: Apply the transmissibility multiplier to all connections between regions I and J
   - NOAQUNNC: Do not apply the transmissibility multiplier to numerical aquifer connections.
   - DEFAULT: ALL

The set of records must end with a single line containing only a slash (/).

The total number of gradients for inter-region transmissibility parameters as well as fault transmissibility parameters (specified using keyword **HMFAULTS**) must not exceed **MFHIST** specified in item 4 of keyword **HMDIMS**.

See also keywords **MULTREGT** and **HMMMREGT**.

**Example**

```
HMMULRGT
  1 2 /
  3 4 /
```
HMMULTxx **Cumulative multipliers for gradient parameters**

These keywords set cumulative pore volume or transmissibility multipliers for Gradient runs.

- **HMMULTX** Cumulative X-transmissibility multiplier
- **HMMULTY** Cumulative Y-transmissibility multiplier
- **HMMULTZ** Cumulative Z-transmissibility multiplier
- **HMMLTXY** Cumulative X and Y transmissibility multiplier
- **HMMULTPV** Cumulative pore volume multiplier

These keywords are analogous to MULTPV and MULTX, MULTY, MULTZ etc. except that HMMULTX etc. act as a multiplier on any values of MULTX that have been previously specified in either the GRID or EDIT sections. It is cumulative.

For modifications to permeability parameters, the following keywords are also recognized only in the GRID section:

- **HMMLTPX** X-permeability multiplier
- **HMMLTPY** Y-permeability multiplier
- **HMMLTPZ** Z-permeability multiplier
- **HMMLTPXY** XY-permeability multiplier

The correspondence with the MULTx keywords is shown below:

<table>
<thead>
<tr>
<th>Cartesian</th>
<th>Radial alias</th>
</tr>
</thead>
<tbody>
<tr>
<td>MULTX / HMMULTX</td>
<td>MULTR / HMMULTR</td>
</tr>
<tr>
<td>MULTY / HMMULTY</td>
<td>MULTTHT / HMMULTTH</td>
</tr>
<tr>
<td>MULTZ / HMMULTZ</td>
<td>MULTZ / HMMULTZ</td>
</tr>
<tr>
<td>MULTPV / HMMULTPV</td>
<td>MULTPV / HMMULTPV</td>
</tr>
</tbody>
</table>

*Table 3.15: Correspondence between MULTxx and HMMULTxx keywords*

Additionally, the keyword **HMMLTXY** (HMMLTRTH in radial geometry) specifies both HMMULTX and HMMULTY.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The values specified for HMMULTX act as cumulative multipliers on the transmissibilities calculated by ECLIPSE for the +X face of each grid block or specified directly using TRANX in the EDIT section. Thus, a value for HMMULTX specified for block (I,J,K) multiplies the transmissibility between blocks (I,J,K) and (I+1,J,K). The treatment of HMMULTY and HMMULTZ is analogous. Any non-neighbor connections due to faults have transmissibilities that reflect the HMMULTX/Y/Z values.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

The set of records must end with a single line containing only a slash (/). The set of records must be terminated with a blank record containing just a slash.
Each keyword may also be used in conjunction with the BOX and EQUALS keywords. Within the GRID section it is also possible to use them together with the COPYBOX and EQUALREG keywords.

Notes

• Use in conjunction with HMTRANX\textsuperscript{n} etc. keywords

Since the transmissibilities calculated by ECLIPSE for a base dataset (in the absence of any HMMULTX data) depend on existing MULTX values, the derivatives calculated by ECLIPSE for the transmissibility multiplier HMTRANX\textsuperscript{n} are with respect to HMMULTX and not with respect to MULTX. This ensures all the derivatives consistently refer to additional multipliers on the base dataset.

• Use in the GRID section

If any keyword HMMULTX/HMMULTY/HMMULTZ etc. (or HMMULTPV) is entered in the GRID section, then it applies only to the transmissibilities (or pore volumes) calculated by ECLIPSE using the information in the GRID section and not to any transmissibilities entered directly in the EDIT section using the TRANX/TRANY/TRANZ (or PORV) keyword. Any MULTX/Y/Z (MULTPV) values that are not specified when the end of the GRID section is reached default to 1.0.

• Use in the EDIT section

If any keyword HMMULTX/HMMULTY/HMMULTZ etc. (or HMMULTPV) is entered in the EDIT section then it acts as an additional multiplier on the transmissibilities (or pore volumes) after reading any modified transmissibilities (pore volumes) from the EDIT section. Any non-neighbor connections not entered explicitly are modified to reflect the edited values of HMMULTX/HMMULTY/HMMULTZ.

• In ECLIPSE 100 only, use with the Local Grid Refinement option

By default, any HMMULT\textsuperscript{x} data is inherited in local cells from the corresponding global host cells. Alternatively, modifiers can be defined for local cells between REFINE and ENDFIN pairs.

You should be aware that HMMULTX etc. (exactly analogous to MULTX etc.) from the global grid translates into a multiplier between local cells which share a boundary of the global grid to which HMMULTX etc. is applied, and not between local cells which share a boundary completely internal to a global cell. You should define HMMULTX etc. between a REFINE and ENDFIN pair if it is required to apply to all local cells within a box.

Note: The same rules described above in notes "Use in the GRID section" and "Use in the EDIT section" apply to all local grids.

Examples

Example 1

EQUALS
  'MULTX'  0.5  1 10 1 1 1 4 /
  'MULTY'  0.8  1 1 1 5 5 8 /
/

Example 2

After any MULTX/Y/Z keywords
EQUALS

'HMMULTX' 0.4 1 5 1 1 2 3 /
'HMMULTY' 0.7 1 1 2 3 6 7 /
'HMMULTPV' 0.35 4 5 2 5 2 4 /
/

Example 3

EQUALS

'HMMULTPV' 0.35 4 5 2 5 2 4 /
/

Keywords
**HMMULTFT**  
**Modifies the transmissibility across the named fault**

The HMMULTFT keyword can be used to modify the transmissibility across a fault previously defined using the **FAULTS** keyword. It has the same effect as the **MULTFLT** keyword except the multiplier is applied cumulatively.

If **MULTFLT** is not entered for the named fault, then **HMMULTFT** has the same effect as **MULTFLT**. If **MULTFLT** has been previously entered, then **HMMULTFT** multiplies the existing fault transmissibility multiplier.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated after item 2. The remaining item assumes its default value.

The set of records must end with a blank record containing only a slash (/).

1. Fault name (as specified in the **FAULTS** keyword) or fault name root  
   A fault name root, ending with an asterisk (*), can be used to refer to several faults in one record.
2. Transmissibility Multiplier  
   DEFAULT: 1.0
3. Diffusivity Multiplier (used if the Molecular Diffusion option is active)  
   DEFAULT: 1.0

**Notes**

- Use of the **MULTFLT** keyword in the **GRID** section modifies the **MULTX**, **MULTY** and **MULTZ** arrays along the fault trajectory. If the **MULTX**, **MULTY** or **MULTZ** keyword is used for some cells, in addition to the **HMMULTFT** keyword, then the resultant transmissibility multiplier is the product of the two input multipliers.

- If two faults with different names run along the same face of a cell, then both transmissibility multipliers are applied.

See also the **FAULTS** and **RPTGRID** keywords.

**Example**

(See the **FAULTS** keyword example)

```
MULTFLT  -- Multiplier
  '+'zigzag'  0.2  /
/
HMMULTFT  -- Multiplier
  '+'zigzag'  0.8  /
```

**Keywords**

1097
HMMULTSG  Dual porosity sigma factor modifiers

This multiplies the dual porosity sigma factor, analogously to the HMSIGMA keyword in the REGIONS section.

The keyword should be followed by one non-negative real number for every grid block in the current input box.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

The data must be terminated by a slash. The set of records must end with a single line containing only a slash (/).

The keyword may also be used in conjunction with the BOX and EQUALS keywords.

Example

```
EQUALS
  'HMMULTSG' 10.0 1 10 1 1 1 4 /
/
```
HMPROPS  

Section heading for end-point scaling modifiers

This keyword is used to supply modifiers to the end-points by the SimOpt program.

The keyword acts as a section heading, and must occur just before the SOLUTION section (that is at the end of the REGIONS section, or at the end of the PROPS section when there is no REGIONS keyword).

(Clearly, modifiers can only be supplied once the SATNUM values are known from the regions section and after all end-point arrays have been specified in the PROPS section; hence HMPROPS should occur after reading PROPS and REGIONS data.)

HMPROPS directs ECLIPSE to treat ensuing keywords as modifiers to any allowable HMxxxxxx keywords pertaining to end-points. The modifier keywords BOX, EQUALS, COPY, MINVALUE, MAXVALUE and ADD may be used with the HMxxxxxx keywords.

When the keyword to be modified occurs without the ADD, COPY, MINVALUE, or MAXVALUE qualifiers, the initial two letters direct ECLIPSE whether to add or multiply the given array: keywords beginning with HM are multiplied, while those beginning with HA are added.

When using Local Grid Refinement, modifiers can be specified between the REFIN / ENDFIN pairs for local grids. By default, ECLIPSE takes the endpoints from the host global cells unless specified for local cells.

The processed end-points can be output using mnemonics SWCR and SOWCR, for example in the RPTSOL keyword in the SOLUTION section.

The allowable HA array keywords that may occur in the HMPROPS section are of the form HASWCR and HASOWCR for example. Generally, any keyword of the form HAXxxxxxx can be used where xxxxxxx is an allowable saturation end point (such as SWCR or SOWCR).

The allowable HM array keywords that may occur in the HMPROPS section are in the form HMKRW and HMKRG. Generally, any keyword of the form HMxxxxxx can be used where xxxxxxx is an allowable relative permeability end point (such as KRW or KRG). For instance,

HASOWCR  Specifies critical oil-in-water saturation modifiers.
HASOGCR  Specifies critical oil-in-gas saturation modifiers.
HASWL  Specifies connate water saturation modifiers.
HMKRW  Specifies maximum water relative permeability modifiers.
HMKRG  Specifies maximum gas relative permeability modifier.
HMKRO  Specifies maximum oil relative permeability modifier.
HMKRWR  Specifies maximum displacing water relative permeability modifier.
HMKRGR  Specifies maximum displacing gas relative permeability modifier.
HMKRORG  Specifies maximum displacing oil-in-gas relative permeability modifier.
HMKRORW  Specifies maximum displacing oil-in-water relative permeability modifier.
HASWLPC  Specifies modifier on lower water saturation used for water capillary pressure.
HASGLPC  Specifies modifier on lower gas saturation used for gas capillary pressure.
HMPCW  Specifies modifier on water capillary pressure.
HMPCG  Specifies modifier on gas capillary pressure.

Example

```
PROPS
   --Define properties and optional end points
REGIONS
   --Regions data
EQUALS
   'HMSOWCR'  1  1 10 1 10 4 5 /
   'HMSOWCR'  2  1 10 1 10 6 7 /

   --Supply modifiers to SOWCR (these modify values of SOWCR entered in
   --the PROPS section, else add to values specified in the SOF2/SOF3/SWOF
   --tables if SOWCR has not previously been input for the given boxes).
HMPROPS
   --Method (a)
ADD
   'HMSOWCR'  0.05 1 10 1 10 4 5 /
   'HMSOWCR'  0.12 1 10 1 10 6 7 /
/
MULTIPLY
   'HMPKRWR'  1.2 1 10 1 10 4 5 /
   'HMPKWR'  1.5 1 10 1 10 6 7 /
/
   --end method (a)

   --Method (b): equivalent to method (a), but using HAxxxxxx and HMxxxxxx
   --keywords.
BOX
   1 10 1 10 4 5 /
HASOWCR
   2100*0.05 /
ENDBOX
BOX
   1 10 1 10 6 7 /
HASOWCR
   100*0.05 /
HMKRW
   100*1.5 /
ENDBOX
   --Additional data for local grids
REFINE
   'OSEBERG' /
BOX
   1 3 1 4 1 6 /
HMKRO
   72*1.1 /
ENDFIN
   --end method (b)
SOLUTION
   --Solution data
```
**Calculate gradients for rock compressibility**

The keyword can be used to obtain gradients of the solution with respect to multipliers on the rock compressibility entered via keyword `ROCK` in each PVT or saturation region. See "Gradient option" in the *ECLIPSE Technical Description* for further information.

If the `ROCK` data refers to `PVTNUM` regions (see item 3 in keyword `ROCKOPTS`) then the keyword should be followed by up to `NTPVT` records (see keyword `TABDIMS`) defining the gradient calculations for each PVT region.

If the `ROCK` data refers to `SATNUM` regions then the keyword should be followed by up to `NTSFUN` records (see keyword `TABDIMS`) defining the gradient calculations for each saturation region.

The total number of rock gradient parameters specified using either the `HMROCK` or `HMROCKET` keywords must not exceed the maximum specified by item 8 of the `HMDIMS` keyword.

Each record must be terminated by a slash (/). The set of records must end with a single line containing only a slash (/).

Each record consists of two items:

1. PVT or saturation table number
   - This should be in the range 1 to `NTPVT` if `PVTNUM` regions are referenced, or in the range 1 to `NTSFUN` if `SATNUM` regions are referenced.

2. Calculate gradients with respect to the multiplier on the rock compressibility
   - 1 calculate
   - 0 do not calculate
   - DEFAULT: 0

The pore volume is calculated as

\[ PV = PV_{ref} \left( 1 + X + X^2 \right) \]  
\[ X = C \left( p - p_{ref} \right) \]

with

where \( C \) and \( p_{ref} \) are the rock compressibility and reference pressure entered in the `ROCK` keyword.

ECLIPSE calculates derivatives with respect to \( \lambda \), a multiplier on \( C \) (at \( \lambda = 1.0 \)), so that

\[ X = \lambda C \left( p - p_{ref} \right) \]

**Example**

```plaintext
HMROCK
  1 1 /
  2 1 /
/
```
The keyword can be used to obtain gradients of the solution with respect to multipliers on the rock compaction pore volume modifiers entered via keyword ROCKTAB or ROCKTABH in each ROCKNUM region. See "Gradient option" in the ECLIPSE Technical Description for further information.

The pore volume is calculated as a multiplier on the reference pore volume using data supplied in the ROCKTAB or ROCKTABH tables:

\[ PV = PV_{ref} M(p) \]  \hspace{1cm} \text{Eq. 3.85}

where \( p \) is the reservoir pressure. If the OVERBURD keyword has been entered,

\[ p = P_{eff} = P_{fluid} * P_{overburden} \]  \hspace{1cm} \text{Eq. 3.86}

If the STRESS option is selected in item 1 of the ROCKOPTS keyword,

\[ p = P_{eff} = P_{overburden} * P_{fluid} \]  \hspace{1cm} \text{Eq. 3.87}

ECLIPSE calculates gradients with respect to two parameters using the functional form

\[ PV = PV_{ref} M\left(p, \lambda_1 + \lambda_2 \frac{(p-p_{ref})}{p_{dim}}\right) \]  \hspace{1cm} \text{Eq. 3.88}

where \( p_{ref} \) is the reference pivot pressure entered in keyword HMRREF for the ROCKNUM region and \( p_{dim} \) is a reference scale pressure used to make \( \lambda_2 \) dimensionless.

Derivatives are calculated with respect to the dimensionless parameters \( \lambda_1 \) and \( \lambda_2 \).

The keyword should be followed by up to NTROCC (item 2 of keyword ROCKCOMP) records defining the gradient calculations for each rock region. The set of records must end with a single line containing only a slash (/).

The total number of rock gradient parameters specified using either the HMROCK or HMROCKT keywords must not exceed the maximum specified by item 8 of the HMDIMS keyword.

Each record consists of three items followed by a slash:

1. Rock table number
   This should be in the range 1 to NTROCC (item 2 of keyword ROCKCOMP)
2. Calculate gradients with respect to \( \lambda_1 \) (at \( \lambda_1 = 1.0 \))
   1 calculate
   0 do not calculate
3. Calculate gradients with respect to \( \lambda_2 \) (at \( \lambda_2 = 0.0 \))
   1 calculate
   0 do not calculate
Example

```
HMROCKT
2 0 1 /
4 1 0 /
/
```
**HMRREF**

**Reference pressure for rock table modifications**

This keyword must be used to supply reference pressures if any gradients with respect to rock compaction parameters in the `ROCKTAB` or `ROCKTABH` tables are requested, or if any modifications are to be applied to these tables.

The keyword should be followed by `NTROCC` records, each terminated by a slash (/). `NTROCC` refers to the number of rock tables, entered in item 2 of keyword `ROCKCOMP`.

Each record specifies the reference pressures $p_{\text{ref}}$ and $p_{\text{dim}}$ in `HMMROCKT`. Each refers to a separate `ROCKNUM` region and contains the reference pressure for that region. The reference pressures are used to modify the rock tables via the `HMMROCKT` keyword.

Each record consists of two items:

1. $p_{\text{ref}}$
   - UNITS: barsa (METRIC), psia (FIELD), atma (LAB)
   - DEFAULT: Error if not specified
2. $p_{\text{dim}}$
   - UNITS: barsa (METRIC), psia (FIELD), atma (LAB)
   - DEFAULT: Error if not specified

Items $p_{\text{ref}}$ and $p_{\text{dim}}$ are described in the `HMMROCKT` keyword.

**Example**

```plaintext
--With NTROCC=4 and p_eff = p_fluid
HMRREF
449.0 500.0 /
500.0 500.0 /
345.0 500.0 /
400.0 500.0 /
```

---

**Keywords**

1104
**HMWELCON** Specify gradients for well connection parameters

This keyword can be used in runs in which the gradient option is required.

The keyword can be used to obtain gradients of the solution with respect to well connection transmissibility factors and skin factors for well connections that are defined using the **COMPDAT** (or **COMPDATL** or **COMPDATM**) keyword in the **SCHEDULE** section.

The keyword is followed by any number of records, described below, each record terminated by a slash (/). A record can be terminated early using a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. **Well name**
   - This item cannot be defaulted. It should be the name of a single well which is defined in the **SCHEDULE** section. You cannot use a wildcard character to refer to more than one well at a time, and you cannot use a well list (that is, as defined using the keyword **WLIST**).

2. **Grid name**
   - If the well connection is not in a local grid, then you should specify ‘FIELD’ for this item. If the well connection is in a local grid, this item should specify the name of the local grid refinement, and it should correspond to the local grid specified for the well connection in the **SCHEDULE** section.
   - DEFAULT: ‘FIELD’

3. **I - location of the connecting grid block**
   - This specifies the I-location of the well connection on the grid specified in item 2. It should correspond to the I-location of the connection specified in the **SCHEDULE** section.
   - This item cannot be defaulted.

4. **J - location of the connecting grid block**
   - This specifies the J-location of the well connection on the grid specified in item 2. It should correspond to the J-location of the connection specified in the **SCHEDULE** section.
   - This item cannot be defaulted.

5. **K - location of the connecting grid block**
   - This specifies the K-location of the well connection on the grid specified in item 2. It should correspond to the K-location of the connection specified in the **SCHEDULE** section.
   - This item cannot be defaulted.

6. **Flag specifying whether connection transmissibility factor gradients are required.**
   - This item should be set to ‘YES’ if connection transmissibility factor gradients are required, and ‘NO’ otherwise.
   - DEFAULT: ‘NO’

7. **Flag specifying whether skin factor gradients are required.**
   - This item should be set to ‘YES’ if skin factor gradients are required, and ‘NO’ otherwise.
DEFAULT: ‘NO’

The total number of well connection gradients requested (that is, the number of connection transmissibility factor gradients plus the number of skin factor gradients) must not exceed the value specified in item 9 of the RUNSPEC keyword HMDIMS.

Example

The example below requests well connection gradients for connections on three layers for well PROD1 in local grid CARF1. These gradients are for skin factor in layer 1, both connection transmissibility factor and skin factor in layer 2, and just connection transmissibility factor in layer 3. It also requests well connection gradients on two layers for well PROD3 on the global grid. These gradients are for both connection transmissibility factor and skin factor in layer 2, and just connection transmissibility factor in layer 3.

This example assumes that the local grid, wells and connections are defined in the SCHEDULE section. It also assumes that item 9 of the HMDIMS keyword in the RUNSPEC section has been set to at least 7 (that is, the total number of gradients requested).

```
HMWELCON
-- Well   Grid   I  J  K  CTF    Skin
   PROD1  CARF1  6  4  1  'NO'   'YES' /
   PROD1  CARF1  6  4  2  'YES'  'YES' /
   PROD1  CARF1  6  4  3  'YES'  /
   PROD3    1*   4  5  2  'YES'  'YES' /
   PROD3    1*   4  5  3  'YES'  'NO'  /
/
```
**HMWPIMLT**  
Calculate gradients with respect to well productivity index

The keyword can be used to obtain gradients of the solution with respect to the productivity index of a nominated well. Derivatives will be calculated with respect to a multiplier applied to all connection flow rates.

The keyword should be followed by a number of records defining the wells to be used. The number of wells specified must not exceed the number specified by item 6 in the **HMDIMS** keyword.

For a well under BHP control, the derivative calculated by **HMWPIMLT** is equivalent to the derivative with respect to a multiplier on the connection transmissibility factor calculated by ECLIPSE or entered directly using keyword **COMPDAT** (or **COMPDATL**).

For a well under rate control, the derivative calculated by **HMWPIMLT** is equivalent to the derivative with respect to a multiplier on the limiting flow rate.

The keyword should be followed by a number of records to declare wells to add to the list of gradients. Wells declared as special history matching wells using keyword **WCONHIST** should not be included in keyword **HMWPIMLT**.

Each record consists of a well name or well name root. A well name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several wells in one record.

Each record must be terminated with a slash (/).

The set of records must end with a single line containing only a slash (/).

**Example**

```
HMWPIMLT
  PROCUCER /
  'SOUTH*' /
/
```

**Keywords**
Gradient regions

This keyword is used in runs in which the Gradient option is required.

The keyword should be followed by an integer between zero and $NMHSTR$ for every grid block in the current input box to indicate the gradient sub-region associated with the keyword to which the cell belongs. A zero indicates that the cell belongs to no subregion, while a positive number defines the sub-region to which the cell belongs.

For each gradient region keyword, ECLIPSE then computes gradients of the solution with respect to the gradient parameters within each user-defined sub-region possessing active cells.

The number of gradient region keywords specified in the run must not exceed $MHISTM$, and the number of subregions within any gradient region cannot be greater than $NMHSTR$; these are specified in RUNSPEC keyword $HMDIMS$.

The following parameters are currently recognized by ECLIPSE:

- **HMTRANX** X-direction transmissibility multipliers
- **HMTRANY** Y-direction transmissibility multipliers
- **HMTRANZ** Z-direction transmissibility multipliers
- **HMTRNXY** X and Y-direction transmissibility multipliers
- **HMPORVM** Pore volume multipliers
- **HMSIGMA** Dual porosity sigma factor multipliers (*ECLIPSE 100* only)
- **HMPERMX** X-direction permeability multipliers
- **HMPERMY** Y-direction permeability multipliers
- **HMPRMXY** X and Y-direction permeability multipliers
- **HMPERMZ** Z-direction permeability multipliers

In addition, for *ECLIPSE 100* only, it is possible to calculate sensitivities with respect to saturation table end-point parameters. The relevant keywords are of the form $HMzzzzzz$ where $zzzzzz$ is any allowable end-point scaling grid array (dependent on the fluid combination in the run). Allowable values for $HMzzzzzz$ are:

- Critical end-points: $HMSWCR$, $HMSGCR$, $HMSOWCR$, $HMSOGCR$. Derivatives are calculated with respect to $SWCR$, $SGCR$ and forth.
- Water connate end-point: $HMSWL$. Derivatives are calculated with respect to $SWL$.
- End-points used for vertical scaling of the relative permeability: $HMKRW$, $HMKRO$, $HMKRG$, $HMKRWR$, $HMKRGK$, $HMKROK$, $HMKRGK$. Derivatives are calculated with respect to multipliers (scale factors) on $KRW$, $KRO$, $KRG$ and so on. Note that the alternative forms such as $KRWR$ may be used even if the alternative three point scaling is not activated by keyword $SCALECRS$.
- End-points used for vertical scaling of capillary pressure curves: $HMPCW$, $HMPCG$. Derivatives are calculated with respect to a multiplier (scale factor) on $PCW$ and $PCG$.

At present sensitivities cannot be obtained when directional or reversible end-point scaling is present.
For a fuller description of the end-point scaling keywords, see "Saturation table scaling" in the ECLIPSE Technical Description.

Transmissibility sensitivities: use of HMTRANX and related keywords

The keywords applying to the transmissibility modifiers will account for variations in transmissibility for any transmissibilities calculated by ECLIPSE using information given in the GRID section or entered directly in the EDIT section. The contributions to the gradient for non-neighbor connections are calculated for all non-neighbor connections calculated by ECLIPSE in the GRID section or modified in the EDIT section. The contribution from any non-neighbor connections entered explicitly using the NNC keyword in the GRID section is ignored. The behavior is exactly analogous to the treatment on non-neighbor connections using MULTX and related keywords.

Well connection factors are not varied with these keywords because these depend on permeability (unless explicitly specified).

Permeability sensitivities: use of HMPERMX and related keywords

The sensitivity of the solutions calculated by ECLIPSE depend on permeabilities in two ways.

Firstly the transmissibilities calculated from information in the GRID section depend on permeability, but any transmissibilities overwritten in the EDIT section do not contribute to these sensitivities. The contributions to the gradient of transmissibility for non-neighbor connections is accounted for all non-neighbor connections calculated by ECLIPSE in the GRID section or modified in the EDIT section. The contribution from any non-neighbor connections entered explicitly using the NNC keyword in the GRID section is ignored.

Secondly, well connection factors calculated in the SCHEDULE section depend on permeabilities using Peaceman’s equation for example; if the connection factor or permeability thickness is specified in COMPDAT then these terms do not contribute towards the permeability sensitivities.

Dual porosity sensitivities: use of HMSIGMA

The keyword accounts for variation of the matrix-fracture coupling transmissibility whatever the recovery mechanism. Therefore, if the Gravity Drainage Model is active, then the derivatives calculated by ECLIPSE also account for variations in SIGMAGDV (the sigma factor for gravity drainage) as well as for SIGMAV. The keyword HMSIGMA will be ignored if the dual porosity model is inactive (RUNSPEC keyword DUALPORO).

Local grid refinement and coarsening options

By default, cells in local grid systems will be assigned HMxxxxxx values from their parent global grid cells, unless explicitly defined for local cells between a REFINE / ENDFIN pair.

The correct gradients can only be obtained at present when all local grids are run in-place using keyword LGRLOCK in the SCHEDULE section. An error will be reported if this is not the case.

The gradient calculation is compatible with the Grid Coarsening option. You should ensure that the same HMxxxxxx region numbers are assigned to all fine grid cells within a single coarse cell, and that any HMMULTxx modifiers apply equally to all fine grid cells within a single coarse cell.

The gradient calculation is also compatible with the Auto refinement option.

At present, it is not possible to obtain permeability sensitivities with the Local Grid Refinement option.
Examples

Example 1

ECLIPSE 100 only

With NDIVIX=3, NDIVIY=3 and NDIVIZ=3 specified in RUNSPEC keyword DIMENS and MHISTM=1 and NMHSTR=3 specified in RUNSPEC keyword HMDIMS. To set two Z-direction transmissibility multiplier parameters, set the appropriate keywords in the REGIONS section:

```
HMTRANZ
3*0 3*1 3*2 /
HMTRANZ
1 0 0 1 0 0 1 0 0
2 0 0 2 0 0 2 0 0
3 0 0 3 0 0 3 0 0 /
```

Example 2

ECLIPSE 100 only

To specify zones within a local grid;

```
REFINE
'GAMMA' /
EQUALS
'HMPORVM' 2 1 24 1 8 3 3 /    layer 2
'HMPORVM' 1 1 24 1 8 2 2 /    layer 3
'HMPORVM' 1 1 24 1 8 2 2 /    layer 2
'HMPORVM' 2 1 24 1 8 3 3 /    layer 3
'HMPORVM' 1 1 24 1 8 2 2 /    layer 2
'HMSOWCR' 1 1 24 1 8 1 6 /    layers 1 to 6
/
ENDFIN
```

and in the SCHEDULE section, specify;

```
LGRLOCK
'**' /
/
```

Example 3

With NDIVIX=3, NDIVIY=3, NDIVIZ=3, MHISTM=1, and NMHSTR=3, to set two pore volume multiplier parameters, set the appropriate keywords in the REGIONS section:

```
HMPORVM
3*0 3*1 3*2 /
HMPORVM
1 0 0 1 0 0 1 0 0
2 0 0 2 0 0 2 0 0
3 0 0 3 0 0 3 0 0 /
```
HRFIN

Radial grid DRV ratios

The keyword HRFIN can be used to dictate the size ratios of DRV values within radial local grids. The keyword should be placed after the keyword introducing the local grid (RADFIN or RADFIN4) and before the local grid data is terminated with ENDFIN.

HRFIN should be followed by NR–1 values terminated with a slash (/), where NR is the number of cells in the refined grid in the radial direction as specified in keyword RADFIN (item 6) or RADFIN4 (item 8). ECLIPSE sets the DR values for the refined cells in proportion to their respective ratios, scaling them to fit within the host cell and honoring the outer radius specified by OUTRAD.

If HRFIN is not specified, radial refinements are automatically given geometrically (ECLIPSE 100), or logarithmically (ECLIPSE 300), increasing DR values, unless the DR (or DRV) values are specified explicitly.

Local grid refinement is described in "Local Grid Refinement and Coarsening" in the ECLIPSE Technical Description.

Example

Set up DR ratios for 5 radial sectors:

```
INRAD
0.25 /

OUTRAD
1000.0 /

HRFIN
2.0 3.0 4.0 7.0 /
```
HSGRO

Hysteresis residual gas saturation in gas-oil system

This keyword specifies the residual gas saturation in a gas-oil system within each grid cell for the hysteresis saturation tables. This is only required when the ODD3P option is being used and when saturation table end-point scaling has been specified by the ENDScale keyword in the RUNSPEC section.

Each HSGRO keyword should be followed by one real number for each grid cell in the current input box specifying the residual saturation within the grid cell. The saturation should be in the range of 0.0 to 1.0 inclusive. If the value of HSGRO is omitted for any grid cell in a model for which ENDScale has been specified, the residual saturation will be obtained from the critical saturation in the appropriate saturation table.

See "Saturation table scaling" in the ECLIPSE Technical Description for further details.

See also the HSORG, HSORW, HSWRO, HSGRW and HSWRG keywords.

Example

With NDIVIX=3, NDIVIY=4 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS, and no input BOX set:

```
HSGRO
60*0.1/
```
HSGRW

Hysteresis residual gas saturation in gas-water system

This keyword specifies the residual gas saturation in a gas-water system within each grid cell for the hysteresis saturation tables. This is only required when the ODD3P option is being used and when saturation table end-point scaling has been specified by the ENDSCALE keyword in the RUNSPEC section.

Each HSGRW keyword should be followed by one real number for each grid cell in the current input box specifying the residual saturation within the grid cell. The saturation should be in the range of 0.0 to 1.0 inclusive. If the value of HSGRW is omitted for any grid cell in a model for which ENDSCALE has been specified, the residual saturation will be obtained from the critical saturation in the appropriate saturation table.

See "Saturation table scaling" in the ECLIPSE Technical Description for further details.

See also the HSORG, HSGRO, HSORW, HSWRO and HSWRG keywords.

Example

With NDIVIX=3, NDIVIY=4 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS, and no input BOX set:

```
HSGRW
60*0.1/
```
HSORG

Hysteresis residual oil saturation in oil-gas system

This keyword specifies the residual oil saturation in an oil-gas system gas within each grid cell for the hysteresis saturation tables. This is only required when the ODD3P option is being used and when saturation table end-point scaling has been specified by the ENDScale keyword in the RUNSPEC section.

Each HSORG keyword should be followed by one real number for each grid cell in the current input box specifying the residual saturation within the grid cell. The saturation should be in the range of 0.0 to 1.0 inclusive. If the value of HSORG is omitted for any grid cell in a model for which ENDScale has been specified, the residual saturation will be obtained from the critical saturation in the appropriate saturation table.

See "Saturation table scaling" in the ECLIPSE Technical Description for further details.

See also the HSGRO, HSORW, HSWRO, HSGRW and HSWRG keywords.

Example

With NDIVIX=3, NDIVIY=4 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS, and no input BOX set:

```
HSORG
60*0.1/
```

Keywords

ECLIPSE Reference Manual

1114
**HSORW**

**Hysteresis residual oil saturation in oil-water system**

This keyword specifies the residual oil saturation in an oil-water system within each grid cell for the hysteresis saturation tables. This is only required when the **ODD3P** option is being used and when saturation table end-point scaling has been specified by the **ENDSCALE** keyword in the RUNSPEC section.

Each **HSORW** keyword should be followed by one real number for each grid cell in the current input box specifying the residual saturation within the grid cell. The saturation should be in the range of 0.0 to 1.0 inclusive. If the value of **HSORW** is omitted for any grid cell in a model for which **ENDSCALE** has been specified, the residual saturation will be obtained from the critical saturation in the appropriate saturation table.

See "Saturation table scaling" in the *ECLIPSE Technical Description* for further details.

See also the **HSORG**, **HSGRO**, **HSWRO**, **HSGRW** and **HSWRG** keywords.

**Example**

With **NDIVIX=3, NDIVIY=4** and **NDIVIZ=5** specified in RUNSPEC keyword **DIMENS**, and no input BOX set:

```
HSORW
60*0.1/
```
**HSWRG**

**Hysteresis residual water saturation in water-gas system**

This keyword specifies the residual water saturation in a water-gas system within each grid cell for the hysteresis saturation tables. This is only required when the **ODD3P** option is being used and when saturation table end-point scaling has been specified by the **ENDSCALE** keyword in the RUNSPEC section.

Each **HSWRG** keyword should be followed by one real number for each grid cell in the current input box specifying the residual saturation within the grid cell. The saturation should be in the range of 0.0 to 1.0 inclusive. If the value of **HSWRG** is omitted for any grid cell in a model for which **ENDSCALE** has been specified, the residual saturation will be obtained from the critical saturation in the appropriate saturation table.

See "Saturation table scaling" in the *ECLIPSE Technical Description* for further details.

See also the **HSORG**, **HSGRO**, **HSORW**, **HSWRO** and **HSGRW** keywords.

**Example**

With **NDIVIX=3, NDIVIY=4** and **NDIVIZ=5** specified in RUNSPEC keyword **DIMENS**, and no input BOX set:

```
HSWRG
  60*0.1/
```

---

**Keywords**

1116
**HSWRO**

**Hysteresis residual water saturation in water-oil system**

This keyword specifies the residual water saturation in a water-oil system within each grid cell for the hysteresis saturation tables. This is only required when the **ODD3P** option is being used and when saturation table end-point scaling has been specified by the **ENDSCALE** keyword in the RUNSPEC section.

Each **HSWRO** keyword should be followed by one real number for each grid cell in the current input box specifying the residual saturation within the grid cell. The saturation should be in the range of 0.0 to 1.0 inclusive. If the value of **HSWRO** is omitted for any grid cell in a model for which **ENDSCALE** has been specified, the residual saturation will be obtained from the critical saturation in the appropriate saturation table.

See "Saturation table scaling" in the *ECLIPSE Technical Description* for further details.

See also the **HSORG**, **HSGRO**, **HSORW**, **HSGRW** and **HSWRG** keywords.

**Example**

With **NDIVIX=3**, **NDIVIY=4** and **NDIVIZ=5** specified in RUNSPEC keyword **DIMENS**, and no input BOX set:

```plaintext
HSWRO
60*0.1/
```

---

**Keywords**

1117
HWELLS

**Horizontal well completion option**

If this keyword is entered in the **RUNSPEC** section, it is possible to specify directions of penetration other than Z in the in the **WELLCOMP** keyword (item 12) or the **COMPDAT** keyword (item 13).

For the X direction of penetration, the Peaceman formula for the well completion (discussed in “The connection transmissibility factor” in the *ECLIPSE Technical Description*) is still used, but with $K_x$ replacing $K_z$ and the cell dimensions in the y-z directions being used to obtain the equivalent radius. Note that gravity effects are not included.

This keyword has no associated data.

*Note:* This keyword cannot be used in a restart run if it has not already been used in the corresponding base case run.

This keyword is not required in ECLIPSE 100.
HWKRO Scaled (high salinity) water-wet end point oil relative permeabilities

The HWKRO keyword specifies the value of oil relative permeability at the maximum oil saturation within each grid block for scaling the (high salinity) water-wet oil relative permeability saturation tables.

The HWKRO keyword should only be used if oil is present in the model and if the end-point scaling, low salinity and surfactant wettability options have been selected via the RUNSPEC keywords ENDScale, LOWSALT and SURFACTW respectively.

The HWKRO keyword should be followed by one real number for each grid block in the current input box specifying the maximum oil relative permeability. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword HWKRO is omitted for some or all of the grid blocks in a model where it could be specified, the oil relative permeabilities at maximum oil saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```plaintext
-- In the RUNSPEC section :
DIMENS
  1 7 3 /
ENDSCALE
/
-- In the PROPS section :
---------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3
HWKRO
6*0.55
6*0.65
6*0.67 /
```
HWKRORG  Scaled (high salinity) water-wet end point oil relative permeabilities

The HWKRORG keyword specifies the value of oil relative permeability at the critical gas saturation within each grid block for scaling the (high salinity) water-wet oil relative permeability saturation tables.

The HWKRORG keyword should only be used if oil and gas are present in the model and if the end-point scaling and surfactant options have been selected via the RUNSPEC keywords ENDSCALE and SURFACT (or SURFACTW) respectively.

The HWKRORG keyword should be followed by one real number for each grid block in the current input box specifying the oil relative permeability at the critical gas saturation. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword HWKRORG is omitted for some or all of the grid blocks in a model where it could be specified, the oil relative permeabilities at critical gas saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```plaintext
-- In the RUNSPEC section :
DIMENS
 1  7  3 /
ENDSCALE
/
-- In the PROPS section :
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX :
   1   2   7   1   3
HWKRORG
 6*0.8
 6*0.8
 6*0.7 /
```
HWKRORW  Scaled (high salinity) water-wet end point oil relative permeabilities

The HWKRORW keyword specifies the value of oil relative permeability at the critical water saturation within each grid block for scaling the (high salinity) water-wet oil relative permeability saturation tables.

The HWKRORW keyword should only be used if oil and water are present in the model and if the end-point scaling and surfactant options have been selected via the RUNSPEC keywords ENDSCALE and SURFACT (or SURFACTW) respectively.

The HWKRORW keyword should be followed by one real number for each grid block in the current input box specifying the oil relative permeability at the critical water saturation. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword HWKRORW is omitted for some or all of the grid blocks in a model where it could be specified, the oil relative permeabilities at critical water saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```plaintext
-- In the RUNSPEC section :
DIMENS
  1  7  3 /
ENDSCALE
/
-- In the PROPS section :
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3
HWKRORW
  6*0.5
  6*0.6
  6*0.6 /
```
**HWKRW**

**Scaled (high salinity) water-wet end point water relative permeabilities**

The **HWKRW** keyword specifies the value of water relative permeability at the maximum water saturation within each grid block for scaling the (high salinity) water-wet water relative permeability saturation tables.

The **HWKRW** keyword should only be used if water is present in the model and if the end-point scaling and surfactant options have been selected via the **RUNSPEC** keywords **ENDSCALE** and **SURFACT** (or **SURFACTW**) respectively.

The **HWKRW** keyword should be followed by one real number for each grid block in the current input box specifying the maximum water relative permeability. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword **HWKRW** is omitted for some or all of the grid blocks in a model where it could be specified, water relative permeabilities at maximum water saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the **ECLIPSE Technical Description** for details of the end-point scaling procedure.

**Example**

```plaintext
-- In the RUNSPEC section :
DIMENS
   1 7 3 /
ENDSCALE
/
-- In the PROPS section :
------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
   1 1 2 7 1 3
HWKRW
  6*0.55
  6*0.65
  6*0.67 /
```

---

**Keywords**

1122
**HWKRWR**

**Scaled (high salinity) water-wet end point water relative permeabilities**

The **HWKRWR** keyword specifies the value of water relative permeability at the critical oil saturation within each grid block for scaling the (high salinity) water-wet water relative permeability saturation tables.

The **HWKRWR** keyword should only be used if water is present in the model and if the end-point scaling and surfactant options have been selected via the **RUNSPEC** keywords **ENDSCALE** and **SURFACT** (or **SURFACTW**) respectively.

The **HWKRWR** keyword should be followed by one real number for each grid block in the current input box specifying the water relative permeability at the critical oil saturation. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword **HWKRWR** is omitted for some or all of the grid blocks in a model where it could be specified, the water relative permeabilities at critical oil saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the **ECLIPSE Technical Description** for details of the end-point scaling procedure.

**Example**

```plaintext
-- In the RUNSPEC section :
DIMENS
  1  7  3  /
ENDSCALE
/

-- In the PROPS section :
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1  1  2  7  1  3
HWKRWR
  6*0.5
  6*0.6
  6*0.6  /
```

---

**Keywords**

- ECLIPSE 100
- ECLIPSE 300
- SPECIAL
- RUNSPEC
- GRID
- EDIT
- PROPS
- REGIONS
- SOLUTION
- SUMMARY
- SCHEDULE

---

**ECLIPSE Reference Manual**

1123
**HWPCW**

**Scaled (high salinity) water-wet maximum oil-water capillary pressure**

The **HWPCW** keyword specifies the maximum value of oil-water capillary pressure for scaling the (high salinity) water-wet water-oil capillary pressure saturation tables.

The **HWPCW** keyword should only be used if water and oil are present in the model and if the end-point scaling and surfactant options have been selected via the **RUNSPEC** keywords **ENDSCALE** and **SURFACT** (or **SURFACTW**) respectively.

The **HWPCW** keyword should be followed by one real number for each grid block in the current input box specifying the maximum oil-water capillary pressure. The data must be terminated by a slash (/).

**UNITS:** bars (**METRIC**), psi (**FIELD**), atm (**LAB**), atm (**PVT-M**)

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword **HWPCW** is omitted for some or all of the grid blocks in a model where it could be specified, the maximum oil-water capillary pressures at connate water saturation will default to the values defined in the appropriate capillary pressure tables.

See "Saturation table scaling" in the **ECLIPSE Technical Description** for details of the end-point scaling procedure.

**Notes**

- The **HWPCW** keyword should not be used in combination with the J-function option specified by the keywords **JFUNC** or **JFUNCR** because the J-function calculation is used to define the water capillary pressure scaling and any values defined by the input **LPCW** keyword will be ignored. In this case the maximum capillary pressure for each cell is reported in the **INIT** file.

- The **HWPCW** values used to scale the oil-water capillary pressure must be consistent with the capillary pressure values in the tables. In particular both must possess the same sign.

**Example**

```plaintext
-- In the **RUNSPEC** section :
DIMENS
1 7 3 /
ENDSCALE /
-- In the **PROPS** section :
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX 1 1 2 7 1 3
HWPCW
6*10.0
6*12.5
6*15.0 /
```

**Keywords**

1124
**HWSNUM** *(High salinity) water-wet saturation function region numbers*

The keyword should be followed by one integer for every grid block specifying the (high-salinity) water-wet saturation function region to which it belongs. The region number should not be less than 1 or greater than \( \text{NTSFUN} \) (see item 1 of keyword **TABDIMS**). The data must be terminated by a slash (/).

The (high salinity) water-wet saturation function region number specifies the saturation table (input using **SWFN**, **SOF3** and related keywords in the **PROPS** section) to be used to calculate relative permeabilities and capillary pressures in each grid block.

If the **LOWSALT** option has also been specified, the water-wet curves are computed as a weighted average between the low salinity water-wet saturation functions (**LWSNUM** region numbers) and the high salinity water-wet saturation functions (**HWSNUM** region numbers). In this case, the salinity weighting factors are taken from the **LSALTFNC** keyword according to the region numbers specified by the **LSNUM** keyword.

The table number specified here also defines the associated table number for the **SURFADDW** keyword. This specifies the weighting between immiscible water-wet and oil-wet saturation functions as a function of adsorbed surfactant concentration. The **HWSNUM** keyword can be used only if the **SURFACTW** keyword has been entered in the **RUNSPEC** section.

Grid blocks are ordered with the X-axis index cycling fastest, followed by the Y- and Z-axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

---

**Note:** The **HWSNUM** keyword is an alias for the **SURFWNUM** keyword. One of these keywords must be specified if the **SURFACTW** option has been specified.

The **LWSNUM** keyword is an alias for the **LSLTWNUM** keyword. One of these keywords must be specified if the **LOWSALT** and **SURFACTW** options have been specified.

The **LSNUM** keyword is an alias for the **LWSLTNUM** keyword. One of these keywords must be specified if the **LOWSALT** option has been specified.

---

**Example**

With \( \text{NTSFUN}=2 \) specified in keyword **TABDIMS** and \( \text{NDIVIX}=8, \text{NDIVY}=6 \) and \( \text{NDIVZ}=5 \) specified in **RUNSPEC** keyword **DIMENS**:

```
HWSNUM 144*1 96*2 /
```

---

**Keywords**

1125
HWSOGCR  Scaled (high-salinity) water-wet critical oil-in-gas saturations

The HWSOGCR keyword specifies the critical oil in gas saturation (that is, the largest oil saturation for which the oil relative permeability to gas is zero) within each grid block for scaling the (high-salinity) water-wet oil relative permeability to gas saturation tables.

The tabulated oil relative permeability to gas functions are scaled between the critical oil in gas saturation defined by the HWSOGCR keyword and the maximum oil saturation defined in terms of the minimum water and gas saturations using the LSWL and SGL keywords respectively.

The HWSOGCR keyword should only be used if gas is present in the model and if the end-point scaling and surfactant options have been selected via the RUNSPEC keywords ENDScale and SURFACT (or SURFActW) respectively.

The HWSOGCR keyword should be followed by one real number for each grid block in the current input box specifying the critical oil in gas saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword HWSOGCR is omitted for some or all of the grid blocks in a model where it could be specified, the critical oil in gas saturations for the (high salinity) water-wet oil relative permeability to gas calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```plaintext
-----RUNSPEC section
DIMENS
  1 7 3 /
ENDSCALE
/
-----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3
HWSOGCR
  5*0.24 10*0.21 3*0.20 /
```
**HWSOWCR**  
**Scaled (high salinity) water-wet critical oil-in-water saturations**

The HWSOWCR keyword specifies the critical oil in water saturation (that is, the largest oil saturation for which the oil relative permeability to water is zero) within each grid block for scaling the (high-salinity) water-wet oil relative permeability to water saturation tables.

The tabulated oil relative permeability to water functions are scaled between the critical oil in water saturation defined by the HWSOWCR keyword and the maximum oil saturation defined in terms of the minimum water and gas saturations using the LSWL and SGL keywords respectively.

The HWSOWCR keyword should only be used if water is present in the model and if the end-point scaling and surfactant options have been selected via the RUNSPEC keywords ENDScale and SURFAct (or SURFACTW) respectively.

The HWSOWCR keyword should be followed by one real number for each grid block in the current input box specifying the critical oil in water saturation within the grid block. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword HWSOWCR is omitted for some or all of the grid blocks in a model where it could be specified, the critical oil in water saturations for the (high salinity) water-wet oil relative permeability to water calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

**Example**

```plaintext
---RUNSPEC section
DIMENS
1 7 3 /
ENDSCALE
/
---PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
   1 1 2 7 1 3
HWSOWCR
5*0.21 10*0.19 3*0.18 /
```

**Keywords**

ECLIPSE Reference Manual  1127
**HWSWCR**

**Scaled (high salinity) water-wet critical water saturations**

The `HWSWCR` keyword specifies the critical water saturation (that is, the largest water saturation for which the water relative permeability is zero) within each grid block for scaling the high salinity (water-wet) water relative permeability saturation tables.

The tabulated water relative permeability functions are scaled between the critical water saturation defined by the `HWSWCR` keyword and the maximum water saturation defined by the `HWSWU` keyword.

The `HWSWCR` keyword should only be used if water is present in the model and if the end-point scaling and surfactant options have been selected via the RUNSPEC keywords `ENDSCALE` and `SURFACT` (or `SURFACTW`) respectively.

The `HWSWCR` keyword should be followed by one real number for each grid block in the current input box specifying the critical water saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword `HWSWCR` is omitted for some or all of the grid blocks in a model where it could be specified, the critical water saturations for the (high salinity) water-wet water relative permeability calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

**Example**

```plaintext
----RUNSPEC section
DIMENS
1 7 3 /
ENDSCALE
/
----PROPS section
---------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
1 1 2 7 1 3 /
HWSWCR
5*0.22 10*0.17 3*0.19 /
ENDBOX
```

**Keywords**

1128
**HWSWL**

**Scaled (high salinity) water-wet connate water saturations**

The *HWSWL* keyword specifies the connate water saturation (that is, the smallest water saturation in a water saturation function table) within each grid block for scaling the (high salinity) water-wet oil-water capillary pressure and oil relative permeability to water tables.

The tabulated oil-water capillary pressure functions are scaled linearly between the connate water saturation defined using the *HWSWL* keyword and the maximum water saturation defined using the *HWSWU* keyword.

The *HWSWL* keyword is also used for scaling the oil relative permeability curves. If the connate water saturation has additionally been specified using the *HWSWLPC* keyword, this saturation end-point will be used instead for scaling the oil-water capillary pressure curves. The connate water saturation entered using *HWSWL* will then only be used to scale the oil relative permeability curves.

The *HWSWL* keyword should only be used if water is present in the model and if the end-point scaling and surfactant options have been selected via the RUNSPEC keywords *ENDSCALE* and *SURFACT* (or *SURFACTW*) respectively.

The *HWSWL* keyword should be followed by one real number for each grid block in the current input box specifying the connate water saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword *HWSWL* is omitted for some or all of the grid blocks in a model where it could be specified, the connate water saturations for (high salinity) water-wet oil relative permeability and capillary pressure calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE *Technical Description* for details of the end-point scaling procedure.

**Example**

```plaintext
-----RUNSPEC section
DIMENS
  1 7 3 /
ENDSCALE
 / 
-----PROPS section
--------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3
HWSWL
  5*0.22 10*0.17 3*0.19 /
```

**Keywords**

1129
The \textit{HWSWLPC} keyword specifies the connate water saturation (that is, the smallest water saturation in a water saturation function table) within each grid block for scaling the (high salinity) water-wet oil-water capillary pressure tables.

The tabulated oil-water capillary pressure functions are scaled linearly between the connate water saturation defined using the \textit{HWSWLPC} keyword and the maximum water saturation defined using the \textit{HWSWU} keyword.

The \textit{HWSWLPC} keyword is not used for scaling the oil relative permeability curves.

The \textit{HWSWLPC} keyword should only be used if water is present in the model and if the end-point scaling and surfactant options have been selected via the \textit{RUNSPEC} keywords \textit{ENDSCALE} and \textit{SURFACT} (or \textit{SURFACTW}) respectively.

The \textit{HWSWLPC} keyword should be followed by one real number for each grid block in the current input box specifying the connate water saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10\(\times\)0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword \textit{HWSWLPC} is omitted for some or all of the grid blocks in a model where it could be specified, the connate water saturations for the (high salinity) water-wet capillary pressure calculations will default to the values defined by the \textit{HWSWL} keyword.

See "Saturation table scaling" in the \textit{ECLIPSE Technical Description} for details of the end-point scaling procedure.

\textbf{Example}

\begin{verbatim}
----RUNSPEC section
DIMENS
  1 7 3 /
ENDSCALE
/
----PROPS section
HWSWLPC
  7*0.22   7*0.20   7*0.18 /
\end{verbatim}
**HWSWU**

### Scaled high salinity (water-wet) saturation table

#### maximum water saturations

The **HWSWU** keyword specifies the maximum water saturation (that is, the highest water saturation in a water saturation function table) within each grid block for scaling the (high salinity) water-wet oil-water capillary pressure and water relative permeability saturation tables.

The maximum water saturation is used to scale both the oil-water capillary pressure and water relative permeability curves. The tabulated capillary pressure curves are linearly scaled between the connate water saturation defined using the **HWSWL** or **HWSWLPC** keywords and the maximum water saturation defined using the **HWSWU** keyword. The tabulated water relative permeability curves are scaled between the critical water saturation defined using the **HWSWCR** keyword and the maximum water saturation defined using the **HWSWU** keyword.

The **HWSWU** keyword should only be used if water is present in the model and if the end-point scaling and surfactant options have been selected via the **RUNSPEC** keywords **ENDSCALE** and **SURFACT** (or **SURFACTW**) respectively.

The **HWSWU** keyword should be followed by one real number for each grid block in the current input box specifying the maximum water saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword **HWSWU** is omitted for some or all of the grid blocks in a model where it could be specified, the maximum water saturations for the (high salinity) water-wet water relative permeability and capillary pressure calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the *ECLIPSE Technical Description* for details of the end-point scaling procedure.

### Example

```plaintext
-----RUNSPEC section
DIMENS : 1 7 3 /
ENDSCALE /
-----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX 1 1 2 7 1 3
HWSWU 18*1.00 /
```

**Keywords**

1131
Local grid size ratios in x-direction

The keyword HXFIN can be used to dictate the size ratios of each cell in a local grid refinement. It should be placed after the CARFIN keyword introducing the local grid and before the terminating ENDFIN.

HXFIN applies only to Cartesian refinements.

HXFIN should be followed by NX values, where NX is the total number of cells in the refined grid along the X-direction as specified in keyword CARFIN item 8. The values represent the X-direction size ratios for each the refined grid cells.

You may default the size ratios for all the refined cells belonging to a particular host cell. If no ratios are given for a host cell, it is divided up in equal proportions. Each host cell must have the size ratios of its constituent refined cells either all set or all defaulted.

Local grid refinement is described in "Local grid refinement and coarsening" in the ECLIPSE Technical Description.

Example

To divide a host grid of 3 layers into a refined grid of 7 layers, with splitting ratios defined for the middle layer of the host grid:

```
NXFIN
3 2 2 /
HXFIN
3* 1.0 2.0 2* /
```
Sets reference data for output of hydraulic heads

The data comprises reference data that is used if either HYDH or HYDHF is requested in the RPTRST keyword, or one of the SUMMARY section keywords writing out head data is requested. The record consists of some or all of the following data items, and is terminated by a slash (/).

1. The reference depth for hydraulic head calculation.
   
   UNITS: m (METRIC), ft (FIELD), cm (LAB)

2. The density of freshwater. Only needed if HYDHF requested or another freshwater head.
   
   UNITS: kg/m³ (METRIC), lb/ft³ (FIELD), gm/cc (LAB)

3. Do you want to remove the depth terms from the block head calculation? YES or NO?
   
   DEFAULT: NO

Heads are calculated using the formula

\[ H = z_{ref} - z + \left( p_w - p_{atm} \right) / \rho g \]

where

- \( z_{ref} \) is the reference depth (item 1)
- \( z \) is the depth of the grid block (or the well)
- \( p_w \) is the water phase pressure
- \( p_{atm} \) is the atmospheric pressure
- \( \rho \) is the density of water and
- \( g \) is the acceleration due to gravity.

If item 3 is set to YES the two depth terms are removed from the definition. Note that this only applies to the grid block heads output to restart files or the summary section. Well heads can be adjusted by using the WHEDREFD keyword.

Example

```
HYDRHEAD
45.0000 999.0 /
```

Keywords

1133
HYDRO

Define hydrocarbon type

In a run with \( N_c \) components, this keyword defines the component type. This keyword can be used with the \textsc{solid} option for predicting asphaltene precipitation (see "Solid phase" in the \textit{ECLIPSE Technical Description}).

The keyword should be followed by \( N_c \) values. Each value should be one of the following:

\begin{itemize}
\item \textit{N}  Non-hydrocarbon
\item \textit{H}  Hydrocarbon
\item \textit{P}  Paraffin
\item \textit{C}  Cyclic hydrocarbon (naphthene)
\item \textit{A}  Aromatic
\end{itemize}

\textbf{DEFAULT:} \textit{H}

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the \textit{ECLIPSE Technical Description}.

This keyword is used by the multiphase flash. For asphaltene precipitation, the heaviest hydrocarbon should be an aromatic.

\textbf{Example}

For a four-component system with one EoS region:

\begin{verbatim}
HYDRO
H H P A /
\end{verbatim}
Local grid size ratios in y-direction

The keyword HYFIN can be used to dictate the size ratios of each cell in a local grid refinement. It should be placed after the CARFIN keyword introducing the local grid and before the terminating ENDFIN. HYFIN applies only to Cartesian refinements.

HYFIN should be followed by NY values, where NY is the total number of cells in the refined grid along the y-direction as specified in keyword CARFIN item 9. The values represent the y-direction size ratios for each the refined grid cells.

You may default the size ratios for all the refined cells belonging to a particular host cell. If no ratios are given for a host cell, it will be divided up in equal proportions. Each host cell must have the size ratios of its constituent refined cells either all set or all defaulted.

Local grid refinement is described in "Local grid refinement and coarsening" in the ECLIPSE Technical Description.

Example

To divide a host grid of 3 layers into a refined grid of 7 layers, with splitting ratios defined for the middle layer of the host grid:

```
NYFIN
3 2 2 /
HYFIN
3* 1.0 2.0 2* /
```
Enable the relative permeability hysteresis option

This keyword specifies that the hysteresis option for relative permeabilities is to be used. It is now strongly recommended to control this option using the item 5 of the EHYSTR keyword.

The HYKR keyword should be used in conjunction with the HYST or HYSTJ keyword. If this is selected, IMBNUM values must be entered in the REGIONS section. See "Hysteresis" in the ECLIPSE Technical Description for a description of this option.

See also the related keyword NOHYKR.

Keywords

ECLIPSE Reference Manual

1136
HYMOBGDR  Change the way secondary drainage curves are computed in hysteresis runs with soluble gas

This keyword changes the way that the secondary drainage curves are calculated for Killough and Carlson hysteresis options. In runs with dissolved gas it is possible for the gas saturation to drop below the trapped gas saturation by the resolution of the gas. This keyword allows you to optionally adjust the “historical maximum gas saturation” so that the gas becomes mobile again as soon as the gas saturation starts to increase.

See "Hysteresis" in the ECLIPSE Technical Description for a description of this option.

This keyword has no associated data.
Enable cap pressure hysteresis option

This keyword specifies that the hysteresis option for capillary pressures is to be used. It is strongly recommended to control this option using the item 5 of the EHYSTR keyword.

The HYPC keyword should be used in conjunction with the HYST or HYSTJ keywords. If this is selected, IMBNUM values must be entered in the REGIONS section. See "Hysteresis" in the ECLIPSE Technical Description for a description of this option.

See also the related keyword NOHYPC.
Enable hysteresis option

This keyword requests the use of the hysteresis option. It is now strongly recommended to control this request with the HYSTER option in the SATOPTS keyword.

If the HYST keyword is selected, IMBNUM values must be entered in the REGIONS section. This keyword also implies the use of the Killough treatment of hysteresis (alternatively, HYSTJ may be used to select the Jargon treatment).

In ECLIPSE 300 only, the related keywords HYKR and NOHYKR, HYP and NOHYP specify which combination of hysteresis options are to be applied, relative permeability and/or capillary pressure. If the hysteresis option is selected, either HYKR or NOHYKR, and either HYP or NOHYP must be entered as well.

See "Hysteresis" in the ECLIPSE Technical Description for a description of this option.

This keyword has no associated data.
HYSTCHCK  Check consistency between imbibition and drainage endpoints with the hysteresis option

The HYSTCHCK keyword invokes additional checks for the consistency of imbibition and drainage endpoints when the hysteresis option is active (keyword SATOPTS in the RUNSPEC section).

The keyword has no associated data, and is terminated by a slash (/).

See "Hysteresis" in the ECLIPSE Technical Description for further information.

Example

```
HYSTCHCK
/
```
HYSTJ

Enable hysteresis option using Jargon method

This keyword requests the use of the hysteresis option. It is now strongly recommended to control this request with the HYSTER option in the SATOPTS keyword, and to select the Jargon method using the item 2 of the EHYSTR keyword.

If the HYSTJ keyword is selected, IMBNUM values must be entered in the REGIONS section. This keyword also implies the use of the Jargon treatment of hysteresis (alternatively, HYST may be used to select the Killough treatment).

See "Hysteresis" in the ECLIPSE Technical Description for a description of this option.

The related keywords HYKR and NOHYKR, HYPC and NOHYPc specify which combination of hysteresis options are to be applied, that is relative permeability and/or capillary pressure.

This keyword has no associated data.
Enable hysteresis option using Killough method

HYSTK is an alias for keyword HYST. It is now strongly recommended to request the hysteresis option with the HYSTER option in the SATOPTS keyword and to select the Killough method using the item 2 of the EHYSTR keyword.
Local grid size ratios in z-direction

The keyword HZF IN can be used to dictate the size ratios of each cell in a local grid refinement. It should be placed after the CARFIN or RADFIN keyword introducing the local grid and before the terminating ENDFIN.

HZFIN should be followed by NZ values, where NZ is the total number of cells in the refined grid along the Z-direction as specified in keyword CARFIN item 10, or RADFIN item 8. The values represent the Z-direction size ratios for each of the refined grid cells. You may default the size ratios for all the refined cells belonging to a particular host cell. If no ratios are given for a host cell, it will be divided up in equal proportions. Each host cell must have the size ratios of its constituent refined cells either all set or all defaulted.

The form of the input is analogous to HXFIN and HYFIN.
This section contains an alphabetic listing of keywords starting with the letter I.
IHOST

Groups LGRs together in the same process

The keyword is used to assign local grids manually to a process when using domain decomposition in a run containing LGRs. If the LGR is not assigned manually to a process, it will be automatically assigned to the process which has the smallest number of active refined cells already allocated to it. Declaring the largest LGRs first in the GRID section assists this facility.

The IHOST keyword must be placed after all the local grids have been defined in the GRID section (the keyword must appear after the last ENDFIN in the GRID section).

The data consists of a record for each local grid, each containing the following items of data and terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/). The LGRs can be specified in any order. If an LGR does not appear in the IHOST keyword it will be assigned automatically to a process.

1. Local grid name
2. Process number

This should be an integer between 0 and NDMAIN (item 1 in the RUNSPEC section keyword PARALLEL).

positive Assign the LGR to the specified process.

0 ECLIPSE will automatically assign the LGR to a process.

DEFAULT: 0

If any LGRs have been amalgamated using keyword AMALGAM, ECLIPSE will assign the whole amalgamation to the process assigned to the first LGR listed in that amalgamation.

Example

```
IHOST  -- LGR      process
        'NORTH' 1 /
        'SOUTH' 2 /
        'EAST' 1 /
        'SOUTH' 0 / Any available process
        'CENTRAL' 0 / Any available process
/```

Keywords 1145
Imbibition end point relative permeabilities

The keyword IKRG, IKRGR, IKRW, IKRWR, IKRO, IKROG and IKROW should be used to scale the relative permeability for the imbibition process on a block by block basis. The keywords can only be used when both the end point scaling and hysteresis options are in use (see keywords ENDScale and SATOPTS in the RUNSPEC section).

The imbibition tables can be scaled in exactly the same manner as the drainage tables. A full description of the use of each keyword can be found in the Drainage Keyword documentation.

<table>
<thead>
<tr>
<th>Imbibition Keyword</th>
<th>Drainage Keyword</th>
<th>End-point scaled</th>
</tr>
</thead>
<tbody>
<tr>
<td>IKRG</td>
<td>KRG</td>
<td>Maximum gas relative permeability.</td>
</tr>
<tr>
<td>IKRGR</td>
<td>KRGR</td>
<td>Gas relative permeability at residual oil (or water in gas-water systems).</td>
</tr>
<tr>
<td>IKRW</td>
<td>KRW</td>
<td>Maximum water relative permeability.</td>
</tr>
<tr>
<td>IKRWR</td>
<td>KRWR</td>
<td>Water relative permeability at residual oil (or gas in gas-water systems).</td>
</tr>
<tr>
<td>IKRO</td>
<td>KRO</td>
<td>Maximum oil relative permeability.</td>
</tr>
<tr>
<td>IKROG</td>
<td>KROG</td>
<td>Oil relative permeability at the critical gas saturation.</td>
</tr>
<tr>
<td>IKROW</td>
<td>KROW</td>
<td>Oil relative permeability at the critical water saturation.</td>
</tr>
</tbody>
</table>

*Table 3.16: Imbibition relative permeability keywords*
Requests IKU three-phase relative permeabilities

This keyword requests the use of the IKU three-phase relative permeability model. This is applied to all three phases. For the water phase, both water-oil and water-gas tables must be entered using the \texttt{SWF3} keyword (rather than \texttt{SWFN}). For the gas phase, both gas-oil and gas-water tables must be entered using the \texttt{SGF3} keyword (rather than \texttt{SGFN}).

The IKU method can be used with endpoint scaling. In this case the usual critical water and gas saturation values (\texttt{SWCR}, \texttt{SGCR}) are interpreted as critical water and gas in oil. Critical water in gas values are entered with \texttt{SWGCR}; critical gas in water values are entered with \texttt{SGWCR}.

The IKU method combines the two-phase contributions (for example $K_{rgo}$ and $K_{rgw}$ in the case of $K_{rg}$ using a saturation weighting. The look-up for the two contributions is done using a normalized saturation, scaled to a common critical saturation. The critical saturation is an average of that of the individual curves, using a geometric construction in the saturation ternary diagram.

Note that specifying \texttt{SOF3}, \texttt{SWF3} and \texttt{SGF3} with \texttt{RPTPROPS} will output the three-phase relative permeabilities for the oil, water and gas phases, respectively.

At present the IKU relative permeability technique cannot be used with hysteresis or miscibility. Water and gas saturation tables cannot be plotted with GRAF with this option. The IKU model is described in "IKU method" in the \textit{ECLIPSE Technical Description}. 
Imbibition saturation function region numbers

This keyword is used in runs in which the hysteresis option is being used (see the item HYSTER in the SATOPTS keyword). It specifies which saturation table is to be used for each cell for imbibition processes. The usual saturation table numbers specified using the SATNUM keyword are used for drainage processes and equilibration.

The keyword should be followed by one integer for every grid block in the current input box, specifying the saturation function region to which it belongs. The region number should not be less than 1 or greater than NTSFUN (item 1 in the TABDIMS keyword). The data must be terminated by a slash (/).

The saturation function region number specifies which set of saturation functions (input using either: SGFN, SOF2, SOF3, SOF32D and SWFN, or: SGOF, SLGOF and SWOF in the PROPS section) should be used to calculate relative permeabilities and capillary pressures in each grid block when hysteresis is being used.

If hysteresis is required in only part of the reservoir, the hysteresis model can be switched off in specific grid blocks by setting their IMBNUM numbers equal to their SATNUM numbers.

Further forms of this keyword allow an imbibition saturation table number to be specified for each face of a grid block, when using directional relative permeabilities. These table numbers specify the saturation table numbers used during imbibition processes to calculate relative permeabilities for flows out of a grid block across the six grid block faces (+X, -X, +Y, -Y, +Z, -Z). These forms are:

- IMBNUMX: Imbibition saturation table numbers for the +X face of a grid block
- IMBNUMX-: Imbibition saturation table numbers for the -X face of a grid block
- IMBNUMY: Imbibition saturation table numbers for the +Y face of a grid block
- IMBNUMY-: Imbibition saturation table numbers for the -Y face of a grid block
- IMBNUMZ: Imbibition saturation table numbers for the +Z face of a grid block
- IMBNUMZ-: Imbibition saturation table numbers for the -Z face of a grid block

The saturation tables used during drainage processes are specified with the equivalent form of the KRNUM keyword.

In a radial grid, alternative forms should be used for flows in the radial and azimuthal directions:

- IMBNUMR: Imbibition saturation table numbers for the +R face of a grid block
- IMBNUMR-: Imbibition saturation table numbers for the -R face of a grid block
- IMBNUMT: Imbibition saturation table numbers for the +T face of a grid block
- IMBNUMT-: Imbibition saturation table numbers for the -T face of a grid block

The +T direction is that of increasing azimuthal angle. The IMBNUMZ and IMBNUMZ- keywords are the same in both radial and Cartesian grids.

None of the directional imbibition saturation table keywords (IMBNUM and related keywords) should be used unless DIRECT is selected in RUNSPEC keyword SATOPTS. IMBNUMX, IMBNUMY- and IMBNUMZ- should only be used if IRREVERS is also selected in SATOPTS. If IRREVERS is not selected,
the saturation tables specified IMBNUMX are used for both the +X and -X faces of each grid block. Similarly IMBNUMY is used for the +Y and -Y faces, and IMBNUMZ for the +Z and -Z faces.

If a table number is defaulted for any grid cell, the table number is filled in using the data entered under the IMBNUM keyword.

**ECLIPSE 100**

The imbibition region numbers can be modified during the simulation by re-specifying the IMBNUM keyword in the SCHEDULE section. This procedure should only be carried out with care and is **not** recommended in general.

**ECLIPSE 300**

If the ODD3P relative permeability and capillary pressure hysteresis model has been selected, the IMBNUM keyword should not be used. Instead, the primary, secondary hysteresis and tertiary hysteresis saturation region numbers should be specified using the PSTNUM, ISTNUM and DSTNUM keywords respectively.

In all cases blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2).

Note that spaces must not be inserted on either side of the asterisk.

**ECLIPSE 100**

*Note:* If the IMBNUM keyword is not specified when the hysteresis option has been selected, the imbibition saturation region numbers will be defaulted to 1.

**ECLIPSE 300**

*Note:* If the IMBNUM keyword is not specified when the hysteresis option has been selected, an error condition will be reported and program execution will terminate.

See also the keywords IMBNUMF, EHYSTR and HYSTCHCK.

See "Hysteresis" in the ECLIPSE Technical Description for further information.

**Example**

With NTSFUN=2 specified in keyword TABDIMS; NDIVIX=10, NDIVIY=6 and NDIVIZ=2 specified in RUNSPEC keyword DIMENS; and no input BOX set:

```
IMBNUM
  50*1  70*2 /
```
Matrix-fracture imbibition region numbers

This keyword is optional, and only concerns dual porosity runs (keyword DUALPORO in RUNSPEC) using the hysteresis option (item HYSTER in keyword SATOPTS).

The keyword allows the independent specification of the matrix-fracture imbibition saturation tables. ECLIPSE uses the upstream mobilities to calculate the flows, so

- flow from the matrix to the fracture reflects the imbibition table specified for the matrix (top half of the grid), and
- flow from the fracture to the matrix reflects the imbibition table specified for the fracture (bottom half of the grid).

If a table number is defaulted for any grid cell, the table number is filled in using the data entered under the IMBNUM keyword.

The keyword should be followed by one integer for every grid block in the current input box, specifying its imbibition table region number for matrix-fracture flow. The region number should not be less than 1 or greater than NTSFUN (keyword TABDIMS). The data must be terminated by a slash (/).

In all cases blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

See "Dual porosity” and "Hysteresis" in the ECLIPSE Technical Description for further information.

Example

```plaintext
--- In the RUNSPEC section:
DUALPORO
SATOPTS
'HYSTER' /
TABDIMS
3 /
--- In the REGIONS section:
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 10 2 6 1 4 /
IMBNUMMF
100*3 100*1 /
```
**IMKRVD**

**Imbibition relative permeability end-point versus depth tables**

The different forms of this keyword specify the depth variation of the relative permeability for the imbibition saturation tables. The corresponding drainage relative permeabilities can be set by using the **ENKRVD** keyword.

Seven keywords are provided to allow the relative permeability end-points within each grid cell to be scaled independently. These keywords are:

- **IMKRVD**: Relative permeability end-points for the non-directional saturation function tables
- **IMKRVDX**: Relative permeability end-points for the +X face of each grid cell
- **IMKRVDX−**: Relative permeability end-points for the -X face of each grid cell
- **IMKRVDY**: Relative permeability end-points for the +Y face of each grid cell
- **IMKRVDY−**: Relative permeability end-points for the -Y face of each grid cell
- **IMKRVDZ**: Relative permeability end-points for the +Z face of each grid cell
- **IMKRVDZ−**: Relative permeability end-points for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the relative permeability end-points for flows in the +R, -R, +T, -T, +Z, -Z directions respectively.

The keywords can only be used when both the End Point Scaling and Hysteresis options are in use (see keywords **ENDSCALE** and **SATOPTS** in the RUNSPEC section).

The data comprises **NTENDP** (item 3 of keyword **ENDSCALE**) tables of relative permeability end-points versus depth, one for each end-point scaling region. Each table consists of 8 columns of data, and must be terminated by a slash (/).

1. Depth values.
   - The values should increase monotonically down the column.
   - **UNITS**: m (**METRIC**), ft (**FIELD**), cm (**LAB**)  

2. The corresponding values of the maximum water relative permeabilities.

3. The corresponding values of the maximum gas relative permeabilities.

4. The corresponding values of the maximum oil relative permeabilities.

5. Water relative permeability at the critical oil (or gas) saturation.

6. Gas relative permeability at the critical oil (or water) saturation.

7. Oil relative permeability at the critical gas saturation.

8. Oil relative permeability at the critical water saturation.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than **NSENDP** (item 4 of keyword **ENDSCALE**). If an end-point is defaulted for all depth values, in a problem where a value could correctly be defined, then the end-point for each cell in the same scaling region is the value in the appropriate saturation function table. However if an end-point is defaulted...
at a certain depth but has been defined at other depths, the end-point versus depth table is filled in by linear interpolation within the table and constant extrapolation at the ends of the table. End point values entered for absent phases are ignored.

If both directional and irreversible end-point scaling is requested using switches 'DIRECT' and 'IRREVERS' in ENDSCALE, then the relative permeability end-points for flow across the +X and -X faces of a grid cell are set using the keywords IMKRVDX, IMKRVDX-. Similarly IMKRVDY, IMKRVDY- are used for the +Y, -Y faces and the keywords IMKRVDZ, IMKRVDZ- for the +Z, -Z faces. Scaling of the well connection relative permeability tables is performed using the IMKRVD keyword.

When the end-point scaling is directional and reversible ('DIRECT' but not 'IRREVERS' is specified in ENDSCALE), the keyword IMKRVDX defines the relative permeability end-points for the +X and -X faces of each grid cell. Keywords IMKRVDY, IMKRVDZ are used for both the +Y, -Y faces and the +Z, -Z faces of each grid cell respectively. Scaling of well connection relative permeability tables is performed using the IMKRVD keyword.

If ENDSCALE is specified without either switch 'DIRECT' or 'IRREVERS', then none of the directional forms of the IMKRVD keyword are allowed. The IMKRVD keyword then defines the new relative permeability end-points for flow out of each grid cell face, and the well connection relative permeabilities.

See also keyword ENDNUM in the REGIONS section, and keywords ENKRVD and IMPTVD in the PROPS section.

Example

```plaintext
--- RUNSPEC section
OIL
WATER
GAS
SATOPTS
   HYSSTER /
   ENDSCALE
   -- NTENDP NSENDP
   2* 1* 3 /
   --- PROPS section
   IMKRVD
   ---
   --- Depth   Krw   Krg   Kro    at crit    at crit    at crit    at crit
   ---
   ---
   3000.0   0.50    1.0    1.0     0.2      1*       1*      0.34
   6000.0   0.58    1.0    1.0     0.22     1*       1*      0.36
   9000.0   0.66    0.95   0.95    0.25     1*       1*      0.37 /
```
**IMPCVD**

**Imbibition maximum capillary pressure versus depth tables**

This keyword allows you to specify the depth variation of the imbibition maximum capillary pressure for different end-point scaling regions within the reservoir. The keyword can only be used when the end-point scaling option is active (keyword `ENDSCALE` in the RUNSPEC section) together with the hysteresis model (switch 'HYSTER' in RUNSPEC section keyword `SATOPTS`). The corresponding drainage table data is specified using keyword `ENPCVD`.

The data comprises `NTENDP` (item 3 of keyword `ENDSCALE`) tables of maximum capillary pressure end-points versus depth, one for each end-point scaling region. Each table consists of 3 columns of data, and must be terminated by a slash (/).

1. **Depth values.**
   - The values should increase monotonically down the column.
   - **UNITS:** m (METRIC), ft (FIELD), cm (LAB)

2. **The corresponding values of the maximum gas-oil capillary pressure.**
   - **UNITS:** bars (METRIC), psi (FIELD), atm (LAB)

3. **The corresponding values of the maximum water-oil capillary pressure.**
   - **UNITS:** bars (METRIC), psi (FIELD), atm (LAB)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than `NSENDP` (item 4 of keyword `ENDSCALE`). If a maximum capillary pressure is defaulted for all depth values, in a problem where a value could correctly be defined, then the end-point for each cell in the same scaling region is the value in the appropriate saturation function table. However, if a maximum capillary pressure is defaulted and the end-point has been defined at other depths, the end-point versus depth table is filled in by linear interpolation within the table and constant extrapolation at the ends of the table. End point values entered for absent phases are ignored.

See also keyword `ENDNUM` in the REGIONS section, and keyword `ENPCVD` in the PROPS section.

**Example**

```plaintext
---- RUNSPEC section
OIL
WATER
GAS
ENDSCALE
  -- NTENDP  NSENDP
  2* 1 2 /
---- PROPS section
IMPCVD
  3000.0 1.4 23.0
  9000.0 1.2 35.0 /
```

**Keywords**

- IMPCVD
- ENPCVD
- ENPCVD
**Selects the IMPES solution option**

This keyword selects the IMPES Solution option. Under IMPES (IMplicit Pressures, Explicit Saturations), only the cell pressures are solved in coupled form at each iteration, the saturation and composition variables being held constant in the interblock flows. These are then updated at the end of the timestep, once the new pressure values are known.

See "Formulation of the Equations" in the *ECLIPSE Technical Description* for more information.

IMPES contains no items.

- When using ECLIPSE 300, the maximum rate of change of capillary pressure in explicit cells specified by the `DPCDT` keyword, which is assumed to be active by default for the IMPES solution method, should be reviewed and checked for compatibility with the magnitudes of the oil-water and gas-oil capillary pressures specified via the saturation functions.

- The IMPES solution option cannot be used in conjunction with the `PARALLEL` option in ECLIPSE 100.

- The `DIMPES` keyword in the `SCHEDULE` section can be used to dynamically set the IMPES solution procedure in ECLIPSE 100.
**Import data**

This keyword enables values such as porosities and permeabilities to be imported from an INIT file. The keyword contains two records, each terminated by a slash (/).

The first record contains the filename, where the extension of the file specifies the type and format of the file.

There are currently two choices of file name extensions:

- **INIT** binary input
- **FINIT** formatted input (ASCII)

The file must be in a format as given by output from an ECLIPSE simulation.

The second record contains the list of arrays to be read.

The keyword can be used for fast initialization of the grid arrays. This is especially useful when running SECTOR models.

---

**Note:** In order to obtain the correct information from the INIT file, this keyword also requires the grid file. The EGRID file, if present, is used in preference to the GRID file.

---

**Example**

```plaintext
-- Import from OLDCASE.INIT
IMPFILE
  'OLDCASE.INIT' /
  PORO PERMX PERMY PERMZ /
```

---

**Keywords**

1155
Sets the solution procedure to fully implicit

This keyword sets the solution procedure to IMPLICIT for the whole simulation.

Use of the DIMPLICIT keyword in the SCHEDULE section is preferred. Please consult this keyword for more information.

Example

```
  IMPLICIT
```

Keywords

<table>
<thead>
<tr>
<th>x</th>
<th>ECLIPSE 100</th>
<th>ECLIPSE 300</th>
<th>SPECIAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>RUNSPEC</td>
<td>GRID</td>
<td>EDIT</td>
</tr>
<tr>
<td></td>
<td>PROPS</td>
<td>REGIONS</td>
<td>SOLUTION</td>
</tr>
</tbody>
</table>
|   | SUMMARY     | SCHEDULE  

1156
The IMPORT keyword specifies the name of the binary file to be imported from a grid pre-processor at the current position in the data file.

When used in the GRID section, this keyword supplies an equivalent input for data generated using a grid pre-processor, which would otherwise be specified by keywords with associated formatted data.

This keyword can also be used to import binary data from other programs in single or double precision.

The keyword should be followed by the following items of data:

1. Name of import file
   The file name should not exceed 72 characters in length.

2. Formatted flag
   There are two choices:
   
   FORMATTED    formatted file
   UNFORMATTED   unformatted file

   The word may be shortened to F or U.

   DEFAULT: UNFORMATTED

   The data should be terminated by a slash (/).

See also the GDFILE keyword.

**Example**

This entry causes ECLIPSE to import grid data from the unformatted file ROCK.GRDBIN. At the end of that file, ECLIPSE switches back to the next keyword in the current file.

```
IMPORT
ROCK.GRDBIN /
```
The IMPSAT solution option

This keyword selects the IMPSAT solution option. This treats pressure and phase saturations implicitly, but phase compositions explicitly. This method is ideal for runs involving large capillary pressures, which would cause instability in IMPES or AIM methods.
**Imbibition end-point versus depth tables**

The different forms of this keyword specify the depth variation of the imbibition table end-points for different end-point scaling regions within the reservoir. The corresponding drainage table end-points are specified using the `ENPTVD` keyword.

Seven keywords are provided to allow the saturation end-points within each grid cell to be scaled independently. These keywords are:

- **IMPTVD**: Saturation end-points for the non-directional saturation function tables
- **IMPTVDX**: Saturation end-points for the +X face of each grid cell
- **IMPTVDX-**: Saturation end-points for the -X face of each grid cell
- **IMPTVDY**: Saturation end-points for the +Y face of each grid cell
- **IMPTVDY-**: Saturation end-points for the -Y face of each grid cell
- **IMPTVDZ**: Saturation end-points for the +Z face of each grid cell
- **IMPTVDZ-**: Saturation end-points for the -Z face of each grid cell

In radial grids, the same directional keywords should be used to represent the saturation end-points for flows in the +R,-R,+T,-T,+Z,-Z directions respectively.

The keywords can only be used when both the end-point scaling and hysteresis options are in use (see keywords `ENDSCALE` and `SATOPTS` in the `RUNSPEC` section).

The data comprises `NTENDP` (item 3 of keyword `ENDSCALE`) tables of saturation end-points versus depth, one for each end point scaling region. Each table consists of 9 columns of data, and must be terminated by a slash `/`.

1. Depth values.
   The values should increase monotonically down the column.
   
   **UNITS:** m (**METRIC**), ft (**FIELD**), cm (**LAB**)

2. The corresponding values of the connate water saturation.
3. The corresponding values of the critical water saturation.
4. The corresponding values of the maximum water saturation.
5. The corresponding values of the connate gas saturation.
6. The corresponding values of the critical gas saturation.
7. The corresponding values of the maximum gas saturation.
8. The corresponding values of the critical oil-in-water saturation.
9. The corresponding values of the critical oil-in-gas saturation.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than `NSENDP` (item 4 of keyword `ENDSCALE`). Each saturation entry should be in the range 0.0 to 1.0 inclusive. If a saturation end-point is defaulted for all depth values, in a problem where a value could correctly be defined, then the end-point for each cell in the same scaling region is the value in
the appropriate saturation function table. However if a saturation end-point is defaulted and the end-point has been defined at other depths, the end-point versus depth table is filled in by linear interpolation within the table and constant extrapolation at the ends of the table. End point values entered for absent phases are ignored.

If both directional and irreversible end-point scaling is requested using switches 'DIRECT' and 'IRREVERS' in ENDSCALE, then the saturation end-points for flow across the +X and -X faces of a grid cell are set using the keywords IMPTVDX, IMPTVDX-. Similarly IMPTVDY, IMPTVDY- are used for the +Y, -Y faces and the keywords IMPTVDZ, IMPTVDZ- for the +Z, -Z faces. Scaling of the well connection relative permeability tables and capillary pressure functions for equilibration are performed using the IMPTVD keyword.

When the end-point scaling is directional and reversible ('DIRECT' but not 'IRREVERS' is specified in ENDSCALE), the keyword IMPTVDX defines the saturation end-points for the +X and -X faces of each grid cell. Keywords IMPTVDY, IMPTVDZ are used for both the +Y, -Y faces and the +Z, -Z faces of each grid cell respectively. Scaling of well connection relative permeability tables and capillary pressure tables for equilibration are performed using the IMPTVD keyword.

If ENDSCALE is specified without either switch 'DIRECT' or 'IRREVERS', then none of the directional forms of the IMPTVD keyword are allowed. The IMPTVD keyword then defines the new saturation end-points for flow out of each grid cell face, for well connection relative permeabilities and for capillary pressure evaluations in the equilibration algorithm.

See also keyword ENDSPEC in the REGIONS section, and keywords ENPTVD and IMKRVD in the PROPS section.

**Example**

```plaintext
----- RUNSPEC section
OIL
WATER
GAS
SATOPTS
HYSTER /
ENDSCALE
-- NTENDP NSENDP
 2* 1 3 /
--
-- PROPS section
--
-- Modify the imbibition oil to water critical saturation.
--
IMPTVD
3000.0 6* 0.31 1*
9000.0 6* 0.32 1*/
```

**Keywords**

1160
IMSPCVD  Imbibition saturations for $P_c$ curves versus depth tables

This keyword scales the connate water or gas end-point of the imbibition capillary pressure curves without scaling the corresponding oil relative permeability curves. The connate saturations are entered as functions of depth for different end-point scaling regions within the reservoir. The keyword can only be used when the end-point scaling option is active (keyword ENDScale in the RUNSPEC section) together with the hysteresis model (switch 'HYSTER' in RUNSPEC section keyword SATOPTs). The corresponding drainage table data is specified using keyword ENSPCVD.

The data comprises NTENDP (item 3 of keyword ENDScale) tables of connate saturation end-points versus depth, one for each end-point scaling region. Each table consists of 3 columns of data, and must be terminated by a slash (/).

1. Depth values.
   - The values should increase monotonically down the column.
   - UNITS: m (METRIC), ft (FIELD), cm (LAB)

2. The corresponding values of the connate gas saturation.

3. The corresponding values of the connate water saturation.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSENDP (item 4 of keyword ENDScale). If a saturation end-point is defaulted for all depth values, in a problem where a value could correctly be defined, then the end-point for each cell in the same scaling region is the value in the appropriate saturation function table. However if a saturation end-point is defaulted and the end-point has been defined at other depths, the end-point versus depth table is filled in by linear interpolation within the table and constant extrapolation at the ends of the table. End-point values entered for absent phases are ignored.

See also keyword ENDSNum in the REGIONS section, and the keywords ENSPCVD, SWLPC and SGLPC in the PROPS section.

**Example**

```plaintext
----- RUNSPEC section
OIL
WATER
GAS
ENDSCALE
-- NTENDP  NSENDP
  2*1 2 /
----- PROPS section
IMSPCVD
  3000.0 1* 0.21
  9000.0 1* 0.30 /
```

**Keywords**

---
**INCLUDE**

**Include the contents of another named file**

The keyword should be followed by the name of a file from which input is to be taken. This file will be opened, and read to the end of the file unless an **ENDINC** keyword is encountered. The file is then closed, and input resumed from the main file, starting from the next keyword after the **INCLUDE** keyword.

The **INCLUDE** file name may have up to 132 characters.

The data should be terminated by a slash (/).

Nested **INCLUDE** files are possible.

**INCLUDE** files have restrictions when used in the RUNSPEC section.

**Examples**

**Example 1**

This example causes the program to continue input from the file **CASE1.PVI**. At the end of that file, the program switches back to the next keyword in the current file.

```
INCLUDE
CASE1.PVI /
```

**Example 2**

Take input from the **CASE.PVI** file in the Includes directory:

```
INCLUDE
'Includes/CASE1.PVI' /
```
requests output of an INIT file

The INIT file contains a summary of data entered in the GRID, PROPS and REGIONS sections.

If this keyword is included in the GRID section, the program writes out an Initial Data file, containing grid properties and saturation table data, which can be read into the graphics packages.

The file may be formatted (for transfer between computers with different operating systems, or just for readability), or unformatted (the default). If the FMTOUT keyword is used in the RUNSPEC section, the INIT file will be formatted.

In ECLIPSE 100, if the INIT keyword is used, the INIT file will not contain the VE geometry data, as this can make the INIT file very large.

For ECLIPSE 100, See the FILEPS, and RUNSPEC section keyword INSPEC.

The INIT keyword has no associated data.
**INRAD**

**Inner radius for radial geometry or radial local grid refinements**

The INRAD keyword has two main uses. It can either

- Specify the inner radius of the reservoir if radial geometry is used
  (the RADIAL keyword must be specified in the RUNSPEC section)
  or

- Specify the inner radius of the radial refinement
  (the INRAD keyword must be placed within the appropriate RADFIN / ENDFIN bracket).

In either case, this keyword should be followed by a single positive real number defining the inner radius, and a slash (/).

**UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

**DEFAULT:** 0.5 ft = 0.1524 m

INRAD is required only if a radial system is specified using block centered geometry.

**Notes**

Another way to specify the inner radius of a radial refinement is to use the RADFIN keyword. item 10 of the RADFIN keyword can be used to specify this value. If this value is defaulted, or if you wish to overwrite a previously specified value, then the INRAD keyword can be used. The INRAD keyword should be placed within the RADFIN / ENDFIN bracket so that the refinement will use this data accordingly.

See also "Defining radial local grid refinements" in the ECLIPSE Technical Description.

**Example**

```
INRAD
0.2 /  
```
Request an initial index file

This keyword requests that an Initial Index file should be written to describe the arrays in the INIT file. By default, an Initial Index file is always written, but this can be disabled by using the NOINSPEC keyword.

See "File handling in ECLIPSE" in the ECLIPSE Technical Description.

The keyword has no associated data.
INTPC

Invokes the Integrated Pc curve option for dual porosity

This keyword can be used to activate the integrated capillary pressure option in dual porosity runs. The keyword should only be used in dual porosity runs with the gravity drainage option active (keywords DUALPORO and GRAVDR or GRAVDRM in the RUNSPEC section). The integrated $P_c$ option modifies the input capillary pressure curves to improve the prediction of final oil recovery from a matrix block.

The keyword should be followed by one item of data terminated by a slash (/).

1. **Phase flag:**
   - **WATER** requests that the water-oil capillary pressure curves should be modified.
   - **GAS** requests that the gas-oil capillary pressure curves should be modified.
   - **BOTH** requests that both the water-oil and gas-oil capillary pressure curves should be modified.

   **DEFAULT:** BOTH

If the **INTPC** keyword is not present, the curves are not modified.

See "Dual porosity" in the ECLIPSE Technical Description for a description of the integrated capillary pressure option.

**Note:** The **INTPC** option may now be used with the alternative gravity drainage model (GRAVDRM in the RUNSPEC section), as a new pseudoization specific to the GRAVDRM model is now automatically provided if that model is used. In previous versions this use was not advised as the new pseudoization was not available. In the event that unadvised use of **INTPC** with GRAVDRM was made with pre-2007.1 versions, its behavior can be restored with item 107 in OPTIONS (ECLIPSE 100) or item 126 in OPTIONS3 (ECLIPSE 300).

### Example

```
INTPC
BOTH /
```
IONROCK

**Rock ion exchange capacity**

This keyword may only be used when the ECLIPSE 100 multi-component brine model (keyword `BRINE` in conjunction with keyword `ECLMC` in the RUNSPEC section) is active.

The `IONROCK` keyword defines the ion exchange capacity of the rock. This will be used in the cation exchange calculation. Refer also to keywords `IONXSURF` and `IONXROCK`.

See "Brine tracking" in the *ECLIPSE Technical Description* for details of this option.

The keyword should be followed by one non-negative real number for every grid block in the current input box, specifying the rock ion exchange capacities. The data must be terminated by a slash (`/`).

The ion exchange capacity is entered in moles per unit pore volume.

**UNITS:** kg-M/m³ (METRIC), lb-M/ft³ (FIELD), gm-M/cc (LAB)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example `115*0.02`). Note that spaces must not be inserted on either side of the asterisk.

### Example

```
IONROCK
200*0.05 /
```
**IONXROCK**  \[ \text{Rock ion exchange constant} \]

This keyword may only be used when the ECLIPSE 100 multi-component brine model (keyword `BRINE` in conjunction with keyword `ECLMC` in the RUNSPEC section) is active.

The `IONXROCK` keyword activates ion exchange on rock surfaces and defines the ion exchange constant, \( \beta^r \), in the exchange equilibrium expression

\[
\frac{(C_{++})(C_{+}^r)^2}{(C_{++}^r)(C_{+})^2} = \beta^r q^r \tag{Eq. 3.89}
\]

where

- \( C_{++} \) denotes the divalent cation concentration
- \( C_{+} \) denotes the monovalent cation concentration
- \( C_{++}^r \) denotes the divalent cation concentration associated with the rock
- \( C_{+}^r \) denotes the monovalent cation concentration associated with the rock
- \( q^r \) denotes the rock ion exchange capacity
- \( \beta^r \) denotes the ion exchange constant

See "Brine tracking" in the *ECLIPSE Technical Description* for details of this option. `IONXROCK` must be used together with keyword `IONROCK` which defines the rock ion exchange capacity.

The keyword should be followed by NTSFUN records of data (see item 1 in keyword `TABDIMS`), each containing a single non-negative real number. Each record must be terminated by a slash (/).

**UNITS:** (kg-M/rm^3)^(-1) (METRIC), (lb-M/rb)^(-1) (FIELD), (gm-M/rcc)^(-1) (LAB)

**DEFAULT:** 0.0 (exchange equilibrium ignored)

**Example**

For NTSFUN=2:

```
IONXROCK
-- beta
  0.25 /
  0.35 /
```
IONXSURF

**Surfactant ion exchange constant**

This keyword may only be used when the ECLIPSE 100 multi-component brine model (keyword BRINE in conjunction with keyword ECLMC in the RUNSPEC section) is active in conjunction with the Surfactant option (keyword SURFACT in the RUNSPEC section).

The IONXSURF keyword activates ion exchange on surfactant micelles and defines the surfactant equivalent molecular weight and the ion exchange constant, $\beta^s$, in the exchange equilibrium expression

$$\frac{(C_{++})(C^+)^2}{(C_{++}^s)(C^+_s)^2} = \beta^s C^s$$

Eq. 3.90

where

- $C_{++}$ denotes the divalent cation concentration
- $C_+$ denotes the monovalent cation concentration
- $C_{++}^s$ denotes the divalent cation concentration associated with surfactant
- $C^+_s$ denotes the monovalent cation concentration associated with surfactant
- $C^s$ denotes the surfactant concentration
- $\beta^s$ denotes the ion exchange constant

See "Brine tracking" in the ECLIPSE Technical Description for details of this option.

The keyword should be followed by NTSFUN records of data (see item 1 in keyword TABDIMS), each containing two non-negative real numbers. Each record must be terminated by a slash (/).

1. Equivalent molecular weight of surfactant.
   - UNITS: kg/kg-M (METRIC), lb/lb-M (FIELD), gm/gm-M (LAB)
   - DEFAULT: no default
2. Ion exchange constant.
   - UNITS: (kg-M/rm$^3$)$^{-1}$ (METRIC), (lb-M/rb)$^{-1}$ (FIELD), (gm-M/rcc)$^{-1}$ (LAB)
   - DEFAULT: 0.0 (exchange equilibrium ignored)

**Note:** The equivalent molecular weight refers to the weight of surfactant per mole of “active sites”. A surfactant molecule may have more than one active anionic site. For example, a divalent surfactant with a molecular weight of 500 kg/kg-M has an equivalent weight of 250 kg/kg-M.

**Example**

For NTSFUN=2:

```
IONXSURF
  -- Mw  beta
```

Keywords
<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>500</td>
<td>0.25</td>
</tr>
<tr>
<td>435</td>
<td>0.35</td>
</tr>
</tbody>
</table>
**Imbibition table scaled capillary pressure end-points**

These keywords scale the imbibition capillary pressure curves on a block by block basis. The keywords can only be used when both the end-point scaling and hysteresis options are in use (see keywords `ENDSCALE` and `SATOPTS` in the `RUNSPEC` section).

The imbibition tables can be scaled in exactly the same manner as the drainage tables. A full description of the use of each keyword can be found in the documentation of the equivalent drainage keyword, shown in the following table.

<table>
<thead>
<tr>
<th>Imbibition keyword</th>
<th>Drainage keyword</th>
<th>End-point scaled</th>
</tr>
</thead>
<tbody>
<tr>
<td>IPCW</td>
<td>PCW</td>
<td>Maximum water capillary pressures</td>
</tr>
<tr>
<td>IPCG</td>
<td>PCG</td>
<td>Maximum gas capillary pressures</td>
</tr>
</tbody>
</table>

*Table 3.17: Imbibition capillary pressure keywords*
Run is gas condensate

This optional keyword states that the run is a gas condensate. The keyword has three effects:

- Single phase hydrocarbon blocks are taken as gas. See also FACTLI.
- The flash is simplified. When testing the stability of a single phase fluid, it is usual to do two tests, the first assuming the single phase is liquid, and the second assuming the single phase is vapor. However, when this keyword is used, only one test is performed, assuming the single phase fluid is a vapor. This simplification is justified if the fluid is not near the bubble point.
- Since single phase oil states cannot exist, there is no possibility of direct oil to gas transitions, and the near critical interpolation of $K_{ro}$ and $K_{rg}$ is not performed. (This interpolation is described in “Near critical oil and gas relative permeabilities” in the ECLIPSE Technical Description). See also NOMIX.

ISGAS contains no items.

Note: ISGAS should not be used if the fluid is close to the bubble point. Otherwise there will be a sudden change in the vapor fraction from near zero to one as the pressure increases from just below the bubble point pressure to just above the bubble point pressure. This will cause problems in the simulation (unless the fluid is also near the critical point when the vapor and liquid properties are similar). Furthermore, the simplification in the flash calculation is no longer justified and may lead to some two phase states being incorrectly identified as one phase.
Secondary hysteresis saturation function region numbers

The keyword should be followed by one integer for every grid block in the current input box specifying the secondary hysteresis saturation function region to which it belongs. This keyword is used together with \texttt{PSTNUM} and \texttt{DSTNUM} if the \texttt{ODD3P} relative permeability and capillary pressure hysteresis model has been selected in the \texttt{PROPS} section. The region number should not be less than 1 or greater than \texttt{NTSFUN} (see keyword \texttt{TABDIMS}). The data must be terminated by a slash (/).

The secondary hysteresis saturation function region number specifies which set of saturation functions (input using \texttt{SOF3}, \texttt{SGF3}, and \texttt{SWF3} in the \texttt{PROPS} section) should be used to calculate the secondary hysteresis relative permeabilities and capillary pressures in each grid block.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

For grid blocks for which the secondary hysteresis saturation function region numbers are not specified, the values will be assigned from those defined for the primary saturation function region numbers using the \texttt{PSTNUM} keyword. The \texttt{ODD3P} model is implicitly hysteretic. However, hysteresis may be deselected on a per grid block basis by setting the secondary and tertiary hysteresis saturation region numbers equal to the primary saturation region numbers.

See also the saturation function keywords \texttt{SOF3}, \texttt{SGF3} and \texttt{SWF3} in the \texttt{PROPS} section and the primary and tertiary hysteresis saturation region numbers \texttt{PSTNUM} and \texttt{DSTNUM} in the \texttt{REGIONS} section.

Example

With \texttt{NTSFUN=2} specified in keyword \texttt{TABDIMS}; \texttt{NDIVIX=8}, \texttt{NDIVIY=6} and \texttt{NDIVIZ=6} specified in \texttt{RUNSPEC} keyword \texttt{DIMENS}; and no input \texttt{BOX} set:

```
144*1 96*2 /
```
**ISGL-OWCR**  Imbibition table end-points

The ISGL, ISGLPC, ISGCR, ISGU, ISWL, ISWLPC, ISWCR, ISWU, ISOGCR and ISOWCR keywords scale the imbibition saturation table on a block by block basis. The keywords can only be used when both the end-point scaling and hysteresis options are in use (see keywords **ENDSCALE** and **SATOPTS** in the **RUNSPEC** section).

The imbibition tables can be scaled in exactly the same manner as the drainage tables. A full description of the use of each keyword can be found in the documentation of the equivalent drainage keywords, shown in the following table.

<table>
<thead>
<tr>
<th>Imbibition keyword</th>
<th>Drainage keyword</th>
<th>End-point scaled</th>
</tr>
</thead>
<tbody>
<tr>
<td>ISGL</td>
<td>SGL</td>
<td>Connate gas saturation.</td>
</tr>
<tr>
<td>ISGLPC</td>
<td>SGLPC</td>
<td>Connate gas, $P_c$ curves only. ECLIPSE 100 only</td>
</tr>
<tr>
<td>ISGCR</td>
<td>SGCR</td>
<td>Critical gas saturation.</td>
</tr>
<tr>
<td>ISGU</td>
<td>SGU</td>
<td>Maximum gas saturation.</td>
</tr>
<tr>
<td>ISWL</td>
<td>SWL</td>
<td>Connate water saturation.</td>
</tr>
<tr>
<td>ISWLPC</td>
<td>SWLPC</td>
<td>Connate water, $P_c$ curves only. ECLIPSE 100 only</td>
</tr>
<tr>
<td>ISWCR</td>
<td>SWCR</td>
<td>Critical water saturation.</td>
</tr>
<tr>
<td>ISWU</td>
<td>SWU</td>
<td>Maximum water saturation.</td>
</tr>
<tr>
<td>ISOGCR</td>
<td>SOGCR</td>
<td>Critical oil-to-gas saturation</td>
</tr>
<tr>
<td>ISOWCR</td>
<td>SOWCR</td>
<td>Critical oil-to-water saturation.</td>
</tr>
</tbody>
</table>

*Table 3.18: Imbibition saturation table end-point keywords*
ISOLNUM Identifies extent of each independent reservoir region

The ISOLNUM keyword defines the regions of the reservoir that are not connected by a flow path within the reservoir and should not be used in any other circumstances. The only coupling is by the well facilities (through group control). Once the isolated regions of the reservoir have been identified, ECLIPSE will solve them more efficiently as separate units, whose linear equations in general require different numbers of iterations to converge. (See "Independent reservoir regions" in the ECLIPSE Technical Description.)

The keyword should only be used if NRFREG has been set to a value greater than 1 in the RUNSPEC section keyword REGDIMS.

The keyword should be followed by one integer for each grid cell in the full field, specifying the independent reservoir region number to which it belongs. The data must be terminated by a slash (/).

There must be at least one active cell allocated to each reservoir region between 1 and NRFREG. Any cells given a reservoir region number of 0 or > NRFREG are treated as inactive.

ECLIPSE can be made to output the ISOLNUM array for an existing model by specifying the RPTISOL keyword. The ISOLNUM array is output to the Debug file if there is more than one independent reservoir within the model.

Example

Two layers separated by an inactive layer:

```
ISOLNUM
1200*1
1200*0
1200*2 /
```
This section contains an alphabetic listing of keywords starting with the letter J.
JALS

Linear solver for thermal problems

This keyword controls the JALS linear solver that can dramatically reduce CPU times of thermal runs. The JALS linear solver is the default for thermal simulations. The use of the JALS linear solver is most likely to have a benefit for ‘harder’ problems.

The JALS linear solver employs the constrained pressure residual (CPR) technique in which an approximate pressure system is solved for each outer iteration. The solution of the pressure system is referred to as the inner iteration. The GMRES algorithm is used for both the outer and inner iteration. The GMRES algorithm is similar to the ORTHOMIN algorithm and requires storing a set of search directions.

The JALS solver can be enabled in two modes. In mode 2, the solver is generally more accurate and the simulation should require less linear iterations. However, the cost per linear iteration is higher in mode 2 than in mode 1. Since the total CPU time of a simulation is a product of both the number of linear iterations required and the CPU cost per linear iteration there may be circumstances where the model will run faster in mode 1.

With the JALS solver, the memory location for these search directions can either be the heap or the stack. By default, the storage location is the heap since this ensures a stable executable. Placing the search directions on the stack can yield a performance increase since memory allocation and access to the stack is generally faster. However, if the capacity of the stack is exceeded ECLIPSE will abort. On the Windows system, the size of the stack is fixed. On Linux systems, the stack size can often be configured and set unlimited. Please consult your systems administrator. It is recommended that initial runs are performed using the HEAP options.

By default, the JALS solver uses 20 outer iterations, 4 CPR iterations and no residual check (item 4 of JALS set to “OFF”). For some models, 20 iterations may not be enough. Setting item 4 of this keyword to “ON” will set the same number of iterations, but turn the residual check on so that an extra 20 iterations are allowed in any timestep where this check is triggered. Since this switch is checking that the linearized version of the problem has converged residuals, this forces additional non-linear iterations which may slow down a simulation run.

The keyword should be followed by up to four items, and must be terminated by a slash (/).

1. Switch for JALS solver
   2. Enabled in mode 2
   1. Enabled in mode 1
   0. Off
   DEFAULT: 2

2. Memory location for inner iteration search directions
   HEAP. On the heap
   STACK. On the stack
   DEFAULT: HEAP

3. Memory location for outer iteration search directions
   HEAP. On the heap

Keywords
STACK  On the stack

DEFAULT: HEAP

4. Use of residual check, or switch to previous version of JALS (either 2009.1 or 2008.3)

ON  Residual check turned on

OFF  Residual check turned off

DEFAULT: OFF

Note: It is currently not possible to use the JALS solver in a thermal dual porosity (or dual permeability) run containing local grid refinements.

Example

```
JALS
2  HEAP  HEAP  OFF  /
```
**JFUNC**  

**Activates the Leverett J-function option**

This keyword can be used to activate the Leverett J-function option, which scales the water-oil and/or gas-oil capillary pressure functions according to the grid block porosity and permeability. (See "Leverett J-function" in the *ECLIPSE Technical Description* for further details.)

The keyword may only be used if end-point scaling is switched on using the keyword **ENDSCALE** in the **RUNSPEC** section.

The keyword should be followed by some or all of the following items terminated by a slash (/):

1. J-function flag.
   
   - **WATER**: Requests the J-function option for the water-oil capillary pressure only.
   - **GAS**: Requests the J-function option for the gas-oil capillary pressure only.
   - **BOTH**: Requests the J-function option for both water-oil and gas-oil capillary pressure.

   DEFAULT: BOTH

2. Oil-water surface tension.
   
   **UNITS**: dynes/cm (**METRIC**), dynes/cm (**FIELD**), dynes/cm (**LAB**), dynes/cm (**PVT-M**)
   
   The oil-water surface tension must be supplied if the first data item is set to either WATER or BOTH.

3. Oil-gas surface tension.
   
   **UNITS**: dynes/cm (**METRIC**), dynes/cm (**FIELD**), dynes/cm (**LAB**), dynes/cm (**PVT-M**)
   
   The oil-gas surface tension must be supplied if the first data item is set to either GAS or BOTH.

4. Alternative power for the porosity term ($\alpha$)
   
   DEFAULT: 1/2

5. Alternative power for the permeability term ($\beta$)
   
   DEFAULT: 1/2

6. Permeability Direction
   
   - **XY**: The average of **PERMX** and **PERMY** values
   - **X**: **PERMX** value
   - **Y**: **PERMY** value
   - **Z**: **PERMZ** value

   DEFAULT: XY

The J-function option scales the capillary pressure functions according to the rock porosity and permeability.

The dimensionless J-function is entered in place of the capillary pressure function, specified using either the **SWFN**, **SGFN** or the **SWOF**, **SGOF**, **SLGOF** family of keywords (see information written for the third or fourth column of these keywords).
The capillary pressure is then calculated as follows:

\[ P_c = J(S) \cdot ST \cdot \left( \frac{\text{Poro}}{\text{Perm}} \right)^{1/2} \cdot U_{\text{const}} \]  

Eq. 3.91

where

\( J(S) \) J-function entered in the capillary pressure column of the saturation function tables SWFN or SGFN

ST surface tension - data item 2 or 3

Poro cell porosity

Perm cell permeability

\( U_{\text{const}} \) constant depending on the unit system employed

\[ = 0.318316 \text{ (METRIC pressure: barsa perm: mD)} \]

\[ = 4.61678 \text{ (FIELD pressure: psia perm: mD)} \]

\[ = 0.314153 \text{ (LAB pressure: atma perm: mD)} \]

\[ = 0.314153 \text{ (PVT-M pressure: atma perm: mD)}. \]

If data item 6 is defaulted the permeability is taken as \( K = (K_x + K_y)/2 \), unless the model is a cross section, in which case \( K \) is taken as the appropriate horizontal permeability, or unless it is a one-dimensional model, in which the permeability is taken in the direction of the row or column of cells. In addition, irrespective of the permeability direction specified by item 6:

• if the numbers of cells in the x and y directions are both equal to 1, the z-direction permeability values will be used

• if the number of cells in the x direction only is equal to 1, the y-direction permeability values will be used

• if the number of cells in the y direction only is equal to 1 the x-direction permeabilities will be used.

Note: For the INIT file output, the above equation will define the output under the heading PCW or PCG. It is important to note that the capillary pressure chosen from the input table (SWFN or SGFN) will, by default, be the maximum value. If item 74 in OPTIONS is used, then the minimum value will be used instead.

In the above equation, the scaling factor term is expressed as follows:

\[ \text{Scaling Factor} = ST \cdot \left( \frac{\text{Poro}}{\text{Perm}} \right)^{1/2} \cdot U_{\text{const}} \]  

Eq. 3.92

Note: The scaling factor will be output under the heading PCW or PCG in the RPTPROPS keyword. For ECLIPSE 100 only, the argument ENDPT must also be set in the RPTPROPS keyword.

If data items 4 or 5 are present the capillary pressure will be interpreted as follows:

\[ P_c = J(S) \cdot ST \cdot \left( \frac{\text{Poro}^\alpha}{\text{Perm}^\beta} \right) \cdot U_{\text{const}} \]
If the surface tension is required to be a function of pressure then the pressure dependent capillary pressure option (STOW and STOG keywords) can be used to modify the surface tension. In gas-water cases, the JFUNC keyword should apply to the 'WATER' phase, and the oil-water surface tension is taken to be the gas-water surface tension. The gas-oil surface tension and scaling for gas-oil Pc is ignored. In this case:

\[ P_c = J(S) \cdot S T \cdot \frac{S T(P)}{S T_{ref}} \cdot \left( \frac{Poro}{Perm} \right)^{1/2} \cdot U_{const} \]

Eq. 3.93

where

ST(P) and ST_{ref} are input using the STOG or STOW keywords.

**Note:** If the SWATINIT keyword is used in conjunction with the JFUNC keyword, then the J-function scaling is ignored in regions of the field where the SWATINIT array has been set, and will use PPCWMAX instead if set. If the PCW keyword is used in conjunction with the JFUNC keyword, the input PCW array will be ignored when the J-function selects the water-oil capillary pressure. Similarly, if the PCG keyword is input, the input PCG array is ignored when the J-function selects the gas-oil capillary pressure.

The maximum capillary pressure for each cell is reported to the INIT file. This is shown as PCW or PCG properties in the 3D Viewer.

**Example**

```
JFUNC
WATER 22.0 /
```
**JFUNCR**  
**Activates the Leverett J-function option per saturation table**

This keyword can be used to activate the Leverett J-function option on a per saturation table basis, in order to scale either, both or neither the water-oil and the gas-oil capillary pressure functions according to the grid block porosity and permeability. This is an alternative to and extends the functionality of the `JFUNC` keyword which applies a single set of parameters to the entire reservoir model. (See "Leverett J-function" in the *ECLIPSE Technical Description* for further details.)

The `JFUNCR` keyword can only be used if end-point scaling has been selected using the keyword `ENDSCALE` in the `RUNSPEC` section.

The `JFUNCR` keyword comprises `NTSFUN` tables of data, where `NTSFUN` is defined by item 1 of the keyword `TABDIMS` in the `RUNSPEC` section. Each table consists of one row of data containing some or all of the following items, and terminated by a slash (/):

1. J-function flag.
   - **WATER** Requests the J-function option for the water-oil capillary pressure only.
   - **GAS** Requests the J-function option for the gas-oil capillary pressure only.
   - **BOTH** Requests the J-function option for both water-oil and gas-oil capillary pressure.
   - **NONE** Signifies that the water-oil and gas-oil capillary pressure values entered into the saturation function tables should be used directly without application of the Leverett J-Function scaling.
   
   DEFAULT: **BOTH**

   items 2 to 6 inclusive, which are defined below, are ignored and need not be specified if the first data item is set to **NONE**.

2. Oil-water surface tension.
   - **UNITS:** dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT-M)
   
   The oil-water surface tension must be supplied if the first data item is set to either **WATER** or **BOTH**.

3. Oil-gas surface tension.
   - **UNITS:** dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT-M)
   
   The oil-gas surface tension must be supplied if the first data item is set to either **GAS** or **BOTH**.

4. Power for the porosity term ($\alpha$)
   - **DEFAULT:** 0.5

5. Power for the permeability term ($\beta$)
   - **DEFAULT:** 0.5

6. Permeability direction
   - **XY** The average of `PERMX` and `PERMY` values
   - **X** `PERMX` value

---

**Keywords**
Table 1 must be specified. The contents of tables 2 onwards may be defaulted entirely whereupon the values will be copied from the corresponding values defined in the previous table.

The J-function calculation scales the capillary pressure functions according to the rock porosity and permeability. The dimensionless J-function values are **entered instead of** the capillary pressure values in the saturation function tables, specified using either the SWFN, SGFN or the SWOF, SGOF, SLGOF family of keywords (see the description of capillary pressure defined in the final columns of these keywords).

When the dimensionless J-Function values are specified instead of the capillary pressure values in the saturation function tables, the capillary pressure is calculated according to the expression:

\[
P_c = J(S) \cdot S_f
\]

where the scaling factor \( S_f \) is defined by the expression:

\[
S_f = S_T \cdot \left( \frac{\text{Poro}^\alpha}{\text{Perm}^\beta} \right) \cdot U_{\text{const}}
\]

where:

- **J(S)**  J-function values from the input saturation tables
- **ST**  Surface tension - data item 2 or 3
- **Poro**  Cell porosity
- **Perm**  Cell permeability
- \( \alpha \)  Porosity power
- \( \beta \)  Permeability power
- \( U_{\text{const}} \) is a constant which depends upon the unit system employed
  - = 0.318316  (METRIC pressure: barsa perm: mD)
  - = 4.61678  (FIELD pressure: psia perm: mD)
  - = 0.314153  (LAB pressure: atma perm: mD)
  - = 0.314153  (PVT-M pressure: atma perm: mD).

If data item 6 is defaulted, for a three-dimensional reservoir model, the permeability is calculated as the average of the x and y cell permeability values, that is, \( K = (K_x + K_y)/2 \). If the reservoir model is a cross section, the permeability value \( K \) is obtained from the appropriate horizontal permeability. For a one-dimensional model, the permeability value \( K \) is obtained from the permeability corresponding to the direction of the row or column of cells. In addition, irrespective of the permeability direction specified by item 6:
• if the numbers of cells in the x and y directions are both equal to 1, the z-direction permeability values will be used

• if the number of cells in the x direction only is equal to 1, the y-direction permeability values will be used

• if the number of cells in the y direction only is equal to 1 the x-direction permeabilities will be used.

For models which incorporate directional relative permeabilities, the J-function scaling factors for all directions will be the same and are calculated from the values defined in the JFUNCR tables associated with the SATNUM array. The JFUNCR tables associated with the directional drainage arrays KRNUMX, KRNUMX-, KRNUMY, KRNUMY-, KRNUMZ and KRNUMZ- will be ignored. Similarly for models which incorporate both hysteresis and directional relative permeabilities, the J-Function scaling factors for all directions will be the same and are calculated from the values defined in the JFUNCR tables associated with the IMBNUM array. The JFUNCR tables associated with the directional imbibition arrays IMBNUMX, IMBNUMX-, IMBNUMY, IMBNUMY-, IMBNUMZ and IMBNUMZ- are ignored.

Using the JFUNCR keyword it is possible to specify a combination of regions some of which use J-function values and others of which use capillary pressure values in their associated saturation function tables. Care should be exercised, therefore, in order to ensure that the type of data in the capillary pressure column in each of the saturation function tables matches the corresponding specification for that region in the JFUNCR keyword.

The input values specified in the JFUNCR keyword may be output by including the option JFUNCR with the RPTGRID keyword.

The scaled tables may be inspected and verified upon a per grid block basis by means of the diagnostic output specified via the EPSDEBUG keyword.

Note: For the INIT file output, the above equation will define the output under the heading PCW or PCG. It is important to note that the capillary pressure chosen from the input table (SWFN or SGFN) will, by default, be the maximum value. If item 74 in OPTIONS is used, then the minimum value will be used instead.

Note: The scaling factor will be output under the heading PCW or PCG in the RPTPROPS keyword. For ECLIPSE 100 only, the argument ENDPT must also be set in the RPTPROPS keyword.

ECLIPSE 100

If the surface tension is required to be a function of pressure, the pressure dependent capillary pressure option (STOW and STOG keywords) can be used to modify the surface tension. In gas-water cases, the JFUNCR keyword should apply to the water phase, and the oil-water surface tension is taken to be the gas-water surface tension. The gas-oil surface tension and scaling for gas-oil Pc is ignored. In this case:

\[ P_c = J(S) \cdot ST \cdot \frac{ST(P)}{ST_{ref}} \cdot \left( \frac{\text{Poro}^a}{\text{Perm}^b} \right) \cdot U_{const} \]

Eq. 3.96

where

\[ ST(P) \] and \[ ST_{ref} \] are input using the STOG or STOW keywords.

Note: If the SWATINIT keyword is used in conjunction with the JFUNCR keyword, then the J-Function scaling is ignored in regions of the field where the SWATINIT array has been set, and will use PPCWMAX instead if set. If the PCW keyword is used in conjunction with the JFUNCR keyword, the input PCW array will be
ignored when the J-function selects the water-oil capillary pressure. Similarly, if the PCG keyword is input, the input PCG array is ignored when the J-function selects the gas-oil capillary pressure.

**Example**

With NTSFUN=5:

```plaintext
JFUNCR
 BOTH  22.0 18.0 0.50 0.50 X /
 GAS  1*  17.0 0.52 0.48 Y /
 WATER 23.0 1*  0.54 0.46 Z /
 /                        -- Table 4 copied from table 3
 NONE /                    -- Capillary pressures taken directly from
 /                        -- region No 5 saturation function tables
```

---

**Keywords**

1185
This section contains an alphabetic listing of keywords starting with the letter K.
**Gas relative permeability damage initiation**

The **KRGDI** keyword specifies the molar density of individual components at which damage initiates, see the **KRGDM** keyword.

The data comprises NTSFUN rows of data (see item 1 of the **TABDIMS** keywords) each terminated by a slash (/).

Each row consists of N entries, where N = number of hydrocarbon components + the number of water components.

**UNITS:** kg-M/rm³ (METRIC), lb-M/rb (FIELD), gm-M/rcc (LAB), kg-M/rm³ (PVT-M)

**Example**

With NTSFUN=2, 6 hydrocarbon components and 1 water component. Damage occurring due to the presence of the sixth hydrocarbon component at concentrations greater than 0.5.

```
KRGDI
5*0.0 0.5 0.0 /
5*0.0 0.5 0.0 /
```
**KRGDM**

**Gas relative permeability damage multipliers**

The **KRGDM** keyword enables the gas relative permeability to be modified to account for formation damage. The **KRGDM** data consists of damage factors for each component tabulated against gas relative permeability. The modified relative permeability values are then computed as

\[ K_{rg} = M_d K_{rg}' \]

where

- \( K_{rg}' \) is the relative permeability prior to the damage calculation and
- \( M_d \) is the damage multiplier given by

\[
M_d = \prod_i \frac{1}{1 + D_i(K_{rg}') \times \max(0, Z_i - I_i)}
\]

where the product is taken over all hydrocarbon and water components specified with the **COMPS** and **COMPW** keywords:

- \( D_i \) is the damage factor for a given \( K_{rg}' \) and component, found by interpolating the **KRGDM** data
- \( Z_i \) is the molar concentration of component \( i \) (moles per reservoir volume).

The component molar concentrations can be output in the **SUMMARY** section using the **BMLSC** and **BMWAT** keywords, and in the **SOLUTION** and **SCHEDULE** section using the **MLSC** and **MWAT** mnemonics in the **RPTSTR**, **RPTSL**, **RPTSCHED** keywords.

- \( I_i \) is the initial concentration at which damage occurs for a given component.

  See the **KRGDI** keyword

The damage multiplier \( M_d \) can be written to the restart files by including the **KRGDM** item in the **RPTSTR** keyword.

The data comprises **NTSFUN** tables (see item 1 of keyword **TABDIMS**) each terminated by a slash (/). The tables must have at least 2 but not more than **NSSFUN** rows of data (see item 3 of keyword **TABDIMS**).

Each table consists of \( N \) columns of data, where \( N = 1 + \text{number of hydrocarbon components} + \text{number of water components} \):

- **Column 1**
  - **Gas relative permeability**
  - Values should be between 0 and 1 and should increase monotonically down the column.

- **Columns 2 to COMPS + 1**
  - **Damage factors for hydrocarbon components**
  - **UNITS:** (kg-M/rm\(^3\))\(^{-1}\) (**METRIC**), (lb-M/rb)\(^{-1}\) (**FIELD**), (gm-M/rcc)\(^{-1}\) (**LAB**)

- **Columns COMPS + 2 to COMPS + COMPW + 1**
  - **Damage factors for water components**

---

**Keywords**

1188
UNIT: (kg-m/rm$^3$)$^{-1}$ (METRIC), (lb-M/rb)$^{-1}$ (FIELD), (gm-M/rcc)$^{-1}$ (LAB)

**Note:** The damage factors must have values of zero or greater.

**Note:** The KRGDM keyword is only allowed in compositional simulations.

**Example**

With $\text{NTSFUN}=2$ and $\text{NSSFUN} \geq 4$, 6 hydrocarbon components and 1 water component. Damage occurring due to the presence of the sixth hydrocarbon component.

```
KRGDM
 0.0 5*0.0 0.9  0.0
 0.5 5*0.0 0.8  0.0
 0.9 5*0.0 0.75 0.0
 1.0 5*0.0 0.7  0.0
/
 0.0 5*0.0 0.8  0.0
 0.4 5*0.0 0.7  0.0
 0.7 5*0.0 0.6  0.0
 1.0 5*0.0 0.6  0.0
```
The KRG, KRGR, IKRG and IKRGR keywords allow scaling of the relative permeability of gas within each grid block. KRG scales the \( K_r \) at the maximum gas saturation (typically at connate water), and KRGR scales the relative permeability at residual oil saturation (or critical water in a gas-water run). The keyword can only be used when the end-point scaling option is active (keyword \texttt{ENDSCALE} in the \texttt{RUNSPEC} section).

If the KRGR keyword has not been used, the KRG keyword has the effect of scaling the relative permeability value calculated from the appropriate saturation table after the scaled saturation end-points have been accounted for. Hence:

\[
K_{rg} = K_{rg} (\text{Table}) \frac{\text{KRGR (GridBlock)}}{K_{rgmax} (\text{Table})}
\]

The relative permeability is taken to be the value at either the maximum saturation of the saturation table, or at SGU if this has been specified. This is usually equal to \( 1 - S_{wco} \).

If the KRGR keyword has been used, then the scaling honors the \( K_r \) at the critical saturation (SR) of the displacing phase. When the alternative scaling using three saturation nodes is invoked by keyword \texttt{SCALECRS}, SR is given by

\[
\begin{align*}
\text{SR} &= 1 - \text{SOGCR} - \text{SWL} \text{ in gas/oil or gas/oil/water runs} \\
\text{SR} &= 1 - \text{SWCR} \text{ in gas/water runs}
\end{align*}
\]

where SOGCR, SWL and SWCR are the scaled values. When the alternative scaling method is not employed, only the two nodes SGR and SGU are used to scale the saturation endpoints, and SR is taken to be the transformed value of

\[
\text{SR(Table)} = 1.0 - \text{Sogcr(Table)} - \text{Swo}(\text{Table})
\]

or

\[
\text{SR(Table)} = 1.0 - \text{Swo}(\text{Table}).
\]

Hence the two cases are:

\[
\begin{align*}
\text{SG < SR} & \quad K_{rg} = K_{rg} (\text{Table}) \frac{\text{KRGR (GridBlock)}}{K_{rg}(\text{SR})(\text{Table})} \\
\text{SG > SR} & \quad K_{rg} = \text{KRGR} + \frac{\text{KRG-KRGR}}{K_{rgmax} - K_{rg}(\text{SR})(\text{Table}) - K_{rg}(\text{SR})}
\end{align*}
\]

Fourteen keywords are provided to allow the maximum gas relative permeability to be set for each grid cell independently. These keywords are:

- \texttt{KRG} Max gas \( K_r \) for the non-directional saturation function tables
- \texttt{KRGX} Max gas \( K_r \) for the +X face of each grid cell
- \texttt{KRGX-} Max gas \( K_r \) for the -X face of each grid cell
- \texttt{KRGY} Max gas \( K_r \) for the +Y face of each grid cell
- \texttt{KRGY-} Max gas \( K_r \) for the -Y face of each grid cell
KRGZ  Max gas $K_r$ for the +Z face of each grid cell
KRGZ- Max gas $K_r$ for the -Z face of each grid cell
KRGR  $K_r$ at residual for the non-directional saturation function tables
KRGRX $K_r$ at residual for the +X face of each grid cell
KRGRX- $K_r$ at residual for the -X face of each grid cell
KRGRY $K_r$ at residual for the +Y face of each grid cell
KRGRY- $K_r$ at residual for the -Y face of each grid cell
KRGRZ $K_r$ at residual for the +Z face of each grid cell
KRGRZ- $K_r$ at residual for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the gas $K_r$ values for flows in the +R,-R,+T,-T,+Z,-Z directions respectively.

**Note:** None of the KRG keywords should be used if gas is not present in the model. The keyword ENDSCALE should be specified in the RUNSPEC section.

If both directional and irreversible end-point scaling is requested using switches 'DIRECT' and 'IRREVERS' in ENDSCALE, then the maximum gas $K_r$ values for flow across the +X and -X faces of a grid cell are set using the keywords KRGX, KRGX-. Similarly KRGY, KRGY- are used for the +Y,-Y faces and the keywords KRGRZ, KRGRZ- for the +Z,-Z faces.

When the end-point scaling is directional but reversible ('DIRECT' but not 'IRREVERS' is specified in ENDSCALE), the keyword KRGX defines the maximum gas $K_r$ values for both the +X and -X faces of each grid cell. Keywords KRGY, KRGRZ are used for both the +Y,-Y faces and the +Z,-Z faces of each grid cell respectively.

If ENDSCALE is specified without either switch 'DIRECT' or 'IRREVERS', then none of the directional forms of the KRG keyword are allowed. The KRG keyword then defines the maximum gas $K_r$ value for flow across each grid cell face.

Each KRG keyword should be followed by one real number for each grid block in the current input box, specifying the maximum gas relative permeability value. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If any of the KRG* keywords are omitted in a model where they could be accepted, then the maximum gas $K_r$ value defaults to the value used in the appropriate gas saturation function table.

If pure normalized curves are used where (1.0 - SOGCR - SWL) is equal to SGU, then depending on the scaling option used (see the SCALECRS keyword) the two end-points may be equal. If this is the case both KRG and KRGR apply to the same point, and the KRGR keyword is ignored.
If the Hysteresis Model is active (see keyword SATOPTS), then the KRG, KRGR data scales the drainage curve. The imbibition curve can be scaled by using the equivalent IKRG (IKRGR, IKRGX, IKRGX-, ...) keyword.

See also keywords SGU and ENKRVD in the PROPS section.

Refer to "Saturation Table Scaling" of the ECLIPSE Technical Description for more information.

If the ODD3P relative permeability and capillary pressure hysteresis model is selected, this keyword will be ignored.

**Example**

```
-- In the RUNSPEC section:
DIMENS
  1  7  5  /
ENDSCALE
/

-- In the PROPS section:
KRG
  7*1.0
  7*0.96
  7*0.94
  7*0.90
  7*0.85 /
```
The different forms of this keyword allow a saturation table number to be specified for every face of every grid block. These table numbers specify which sets of saturation functions (input using either: \textit{SGFN}, \textit{SOF2}, \textit{SOF3}, \textit{SOF32D} and \textit{SWFN}, or: \textit{SGOF}, \textit{SLGOF} and \textit{SWOF} in the \texttt{PROPS} section) should be used to calculate relative permeabilities for flows out of each grid block across the six grid block faces (+X, -X, +Y, -Y, +Z or -Z).

- \texttt{KRNUMX} Saturation table numbers for the +X face of each grid block
- \texttt{KRNUMX-} Saturation table numbers for the -X face of each grid block
- \texttt{KRNUMY} Saturation table numbers for the +Y face of each grid block
- \texttt{KRNUMY-} Saturation table numbers for the -Y face of each grid block
- \texttt{KRNUMZ} Saturation table numbers for the +Z face of each grid block
- \texttt{KRNUMZ-} Saturation table numbers for the -Z face of each grid block.

In a radial grid, alternative forms should be used for flows in the radial and azimuthal directions:

- \texttt{KRNUMR} Saturation table numbers for the +R face of each grid block
- \texttt{KRNUMR-} Saturation table numbers for the -R face of each grid block
- \texttt{KRNUMT} Saturation table numbers for the +T face of each grid block
- \texttt{KRNUMT-} Saturation table numbers for the -T face of each grid block.

The +T direction is that of increasing azimuthal angle. The \texttt{KRNUMZ} and \texttt{KRNUMZ-} keywords are the same in both radial and Cartesian grids.

None of these keywords should be used unless directional relative permeabilities have been enabled by the keyword \texttt{SATOPTS}. \texttt{KRNUMX-}, \texttt{KRNUMY-} and \texttt{KRNUMZ-} should be used only if irreversible directional relative permeabilities have been enabled by the keyword \texttt{SATOPTS}. If reversible relative permeability tables are to be employed, the saturation tables specified using \texttt{KRNUMX} are used for both the X+ and X- faces of each grid block. Similarly \texttt{KRNUMY} is used for the Y+ and Y- faces, and \texttt{KRNUMZ} for the Z+ and Z- faces.

The relative permeability is thus looked up in the saturation table for flow across the relevant face, for example \texttt{KRNUMX}.

If a table number is defaulted for any grid cell, the table number is filled in using the data entered under the \texttt{SATNUM} keyword.

\textit{Note:} The potential difference between two cells uses the capillary pressures in the tables addressed by each of the \texttt{KRNUM} keywords. If this is not required, then the capillary pressure data in these tables should be set equal to that in the \texttt{SATNUM} tables.
**ECLIPSE 100 only**

**Note:** The saturation function region numbers can be modified during the simulation by re-specifying the KRNUM keyword in the SCHEDULE section. This procedure should only be carried out with care and is **not** recommended in general.

The keyword should be followed by one integer for every grid block in the current input box, specifying the saturation table number for the appropriate face. The table number should not be less than 1 or greater than NTSFUN (see keyword TABDIMS). The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example, 115*2). Note that spaces must not be inserted on either side of the asterisk.

### Example

```plaintext
--In the RUNSPEC section (allow irreversible relative permeabilities):
SATOPTS
'DIRECT' 'IRREVERS' /
TABDIMS
12 /
--In the REGIONS section :
---------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
12 19 4 9 3 7 /
KRNUMZ-48*1
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
5 5 6 6 7 7 8 8
5 5 6 6 7 7 8 8
9 9 10 10 11 11 12 12
9 9 10 10 11 11 12 12
9 9 10 10 11 11 12 12
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
5 5 6 6 7 7 8 8
5 5 6 6 7 7 8 8
9 9 10 10 11 11 12 12
9 9 10 10 11 11 12 12
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
5 5 6 6 7 7 8 8
5 5 6 6 7 7 8 8
9 9 10 10 11 11 12 12
9 9 10 10 11 11 12 12
9 9 10 10 11 11 12 12
/```

**Keywords**

1194
Matrix-fracture flow saturation tables numbers

This keyword is optional and allows the independent specification of the matrix-fracture saturation tables in dual porosity or dual porosity/dual permeability runs. ECLIPSE uses the upstream mobilities to calculate the flows, so

- flow from the matrix to the fracture reflects the saturation table specified for the matrix (top half of the grid)

and

- flow from the fracture to the matrix reflects the saturation table specified for the fracture (bottom half of the grid).

If a table number is defaulted for any grid cell, the table number is filled in using the data entered under the SATNUM keyword.

The keyword should be followed by one integer for every grid block in the current input box, specifying the saturation table region number for matrix-fracture flow. The region number should not be less than 1 or greater than NTSFUN (item 1 in keyword TABDIMS). The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

See "Dual porosity" in the ECLIPSE Technical Description for further information.

Example

```
-- In the RUNSPEC section:
DUALPORO
TABDIMS
3 /
-- In the REGIONS section:
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  12 19 4 9 1 2 /
KRNUMMF
  48*3
  48*1
 /
```
Oil relative permeability damage initiation

The **KRODI** keyword specifies the molar density of individual components at which damage initiates, see the **KRODM**.

The data comprises NTSFUN rows of data (see item 1 of the **TABDIMS** keyword) each terminated by a slash (/).

Each row consists of N entries, where N = number of hydrocarbon components + the number of water components.

UNITs: kg-M/rm\(^3\) (**METRIC**), lb-M/rb (**FIELD**), gm-M/rcc (**LAB**), kg-M/rm\(^3\) (**PVT-M**)

**Example**

With NTSFUN =2, 6 hydrocarbon components and 1 water component. Damage occurring due to the presence of the sixth hydrocarbon component at concentrations greater than 0.5.

```
KRODI
5*0.0 0.5 0.0 /
5*0.0 0.5 0.0 /
```
The **KRODM** keyword enables the oil relative permeability to be modified to account for formation damage. The **KRODM** data consists of damage factors for each component tabulated against oil relative permeability. The modified relative permeability values are then computed as

\[
K_{ro} = M_d K_{ro}'
\]

where

\[
K_{ro}'
\]

is the relative permeability prior to the damage calculation and

\[
M_d
\]

is the damage multiplier given by

\[
M_d = \prod_{i} \frac{1}{1 + D_i(K_{ro}') \times \max(0, Z_i - I_i)}
\]

where the product is taken over all hydrocarbon and water components specified with the **COMPS** and **COMPW** keywords:

- \(D_i\) is the damage factor for a given \(K_{ro}'\) and component, found by interpolating the **KRODM** data
- \(Z_i\) is the molar concentration of component \(i\) (moles per reservoir volume).

The component molar concentrations can be output in the **SUMMARY** section using the **BMLSC** and **BMWAT** keywords, and in the **SOLUTION** and **SCHEDULE** section using the **MLSC** and **MWAT** mnemonics in the **RPTRST**, **RPTSOL**, **RPTSCHED** keywords.

- \(I_i\) is the initial concentration at which damage occurs for a given component
- See the **KRODI** keyword

The damage multiplier \(M_d\) can be written to the restart files by including the **KRODM** item in the **RPTRST** keyword.

The data comprises **NTSFUN** tables (see item 1 of keyword **TABDIMS**) each terminated by a slash (/). The tables must have at least 2 but not more than **NSSFUN** rows of data (see item 3 of keyword **TABDIMS**). Each table consists of N columns of data, where N = 1 + number of hydrocarbon components + number of water components:

- **Column 1**: Oil relative permeability
  - Values should be between 0 and 1 and should increase monotonically down the column.

- **Columns 2 to COMPS + 1**: Damage factors for hydrocarbon components
  - UNITS: (kg-M/rm³)⁻¹ (**METRIC**), (lb-M/rb)⁻¹ (**FIELD**), (gm-M/rcc)⁻¹ (**LAB**)

- **Columns COMPS + 2 to COMPS + COMPW + 1**: Damage factors for water components
Note: The damage factors must have values of zero or greater

Note: The KRODM keyword is only allowed in compositional simulations.

Example

With NTSFUN=2 and NSSFUN≥4, 6 hydrocarbon components and 1 water component. Damage occurring due to the presence of the sixth hydrocarbon component.

```
KRODM
0.0 0.0 0.9 0.0
0.5 0.0 0.8 0.0
0.9 0.0 0.75 0.0
1.0 0.0 0.7 0.0
/ 0.0 0.0 0.8 0.0
0.4 0.0 0.7 0.0
0.7 0.0 0.6 0.0
1.0 0.0 0.6 0.0
```
KRO-ORG  Scaled end point oil relative permeabilities

The KRO, KRORW, KRORG, IKRO, IKRORW and IKRORG keywords scale the relative permeability of oil within each grid block. KRO scales the K_r at the maximum oil saturation (typically at connate water), KRORW scales the relative permeability at the critical water saturation, and KRORG scales the relative permeability at the critical gas saturation. The keyword can only be used when the end-point scaling option is active (keyword ENDSCALE in the RUNSPEC section).

If the KRORW and KRORG keywords have not been used, the KRO keyword has the effect of scaling the relative permeability value calculated from the appropriate saturation table after the scaled saturation end-points have been accounted for. Hence:

\[ K_{ro} = K_{ro} \left( \frac{K_{ro}}{K_{romax}} \right) \]  
Eq. 3.100

The relative permeability is taken to be the value at either the maximum saturation of the saturation table or at SWL if this has been specified.

If the KRORW or KRORG keywords have been used, then the scaling honors the K_r at the critical saturation (SR) of the displacing phase. When the alternative scaling using three saturation nodes is invoked by keyword SCALECRS, SR is given by

\[ SR = 1 - SWCR - SGRL \] for the KRORW keyword
\[ SR = 1 - SGCR - SWL \] for the KRORG keyword

where SWCR and SGCR etc. are the scaled values. When the alternative scaling method is not employed, only two nodes are used to scale the saturation end-points, and SR is taken as the transformed value of

\[ SR (Table) = 1.0 \cdot \frac{SWCR (Table) - SGCO (Table)}{S_{wco} (Table)} \] for the KRORW keyword, or
\[ SR (Table) = 1.0 \cdot \frac{SGCR (Table) - SWL (Table)}{S_{wco} (Table)} \] for the KRORG keyword.

Hence the two cases are:

\[ SO < SR \quad K_{ro} = K_{ro} \left( \frac{K_{ro}}{K_{romax}} \right) \]  
Eq. 3.101

\[ SO > SR \quad K_{ro} = K_{ro} + \frac{K_{ro}}{K_{romax} - K_{ro} (SR)} \left( K_{ro} (Table) - K_{ro} (SR) \right) \]  
Eq. 3.102

Note: In three-phase cases using the STONE II Model for three-phase relative permeability (see keyword STONE2), the oil relative permeability is a function of the water and gas relative permeabilities, and hence the oil relative permeability is modified by use of the KRW and KRG keywords to scale the water and gas relative permeabilities.

Twenty-one keywords are provided to allow the maximum oil relative permeability to be set for each grid cell independently. These keywords are:

- KRO: Max oil K_r for the non-directional saturation function tables
- KROX: Max oil K_r for the +X face of each grid cell
- KROX-: Max oil K_r for the -X face of each grid cell
KROY       Max oil \( K_r \) for the +Y face of each grid cell
KROY-      Max oil \( K_r \) for the -Y face of each grid cell
KROZ       Max oil \( K_r \) for the +Z face of each grid cell
KROZ-      Max oil \( K_r \) for the -Z face of each grid cell
KRORW KROG  \( K_r \) at critical for the non-directional tables
KRORX KROGX \( K_r \) at critical for the +X face of each grid cell
KRORX- KROGX- \( K_r \) at critical for the -X face of each grid cell
KRORWY KROGY \( K_r \) at critical for the +Y face of each grid cell
KRORY- KROGY- \( K_r \) at critical for the -Y face of each grid cell
KROWZ KROGZ \( K_r \) at critical for the +Z face of each grid cell
KROWZ- KROGZ- \( K_r \) at critical for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the oil \( K_r \) value for flows in the +R,-R,+T,-T,+Z,-Z directions respectively.

None of the KRO keywords should be used if oil is not present in the model. The keyword ENDSCALE should also be set in the RUNSPEC section.

If both directional and irreversible end-point scaling is requested using switches 'DIRECT' and 'IRREVERS' in ENDSCALE, then the maximum oil \( K_r \) values for flow across the +X and -X faces of a grid cell are set using the keywords KROX, KROX-. Similarly KROY, KROY- are used for the +Y,-Y faces and the keywords KROZ, KROZ- for the +Z,-Z faces.

When the end-point scaling is directional but reversible ('DIRECT' but not 'IRREVERS' is specified in ENDSCALE), the keyword KROX defines the maximum oil \( K_r \) values for both the +X and -X faces of each grid cell. Keywords KROY, KROZ are used for both the +Y,-Y faces and the +Z,-Z faces of each grid cell respectively.

If ENDSCALE is specified without either switch 'DIRECT' or 'IRREVERS', then none of the directional forms of the KRO keyword are allowed. The KRO keyword then defines the maximum oil \( K_r \) value for flow across each grid cell face.

Each KRO keyword should be followed by one real number for each grid block in the current input box, specifying the maximum oil relative permeability value. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If any of the KRO* keywords are omitted in a model where they could be accepted, then the maximum oil \( K_r \) value defaults to the value used in the appropriate oil saturation function table.

If pure normalized curves are used where SWCR equals SWL or SGCR equals SGL, then depending on the Scaling option used (see the SCALECRS keyword) the two end-points may be equal. If this is the case, both
KROW and KRORW (or KROG and KROG) apply to the same point and the KRORW (KROG) keyword is ignored.

If the Hysteresis model is active (see keyword SATOPTS), then the KRO, KRORW, KROG data will scale the drainage curve. The imbibition curve can be scaled by using the equivalent IKRO (IKROG, IKROG, IKROG) keyword.

See also keywords SWL and ENKRVD in the PROPS section.

Refer to "Saturation table scaling" in the ECLIPSE Technical Description for more information.

**ECLIPSE 300**

If the ODD3P relative permeability and capillary pressure hysteresis model is selected, this keyword will be ignored.

**Example**

```plaintext
-- In the RUNSPEC section:
DIMENS 10 1 4 /
ENDSCALE /
-- In the PROPS section:
KRO 10*0.9 10*0.98 10*1.0 10*1.0 /
KRORW 10*0.8 10*0.86 10*0.9 10*1.0 /
```
**KRWDI**

Water relative permeability damage initiation

The KRWDI keyword specifies the molar density of individual components at which damage initiates, see the KRWDM.

The data comprises NTSFUN rows of data (see item 1 of the TABDIMS keyword) each terminated by a slash (/).

Each row consists of N entries, where N = number of hydrocarbon components + the number of water components.

UNITS: kg-M³/rm³ (METRIC), lb-M/rb (FIELD), gm-M/rcc (LAB), kg-M/rm³ (PVT-M)

**Example**

With NTSFUN=2, 6 hydrocarbon components and 1 water component. Damage occurring due to the presence of the sixth hydrocarbon component at concentrations greater than 0.5.

```
KRWDI
5*0.0 0.5 0.0 /
5*0.0 0.5 0.0 /
```
Water relative permeability damage multipliers

The KRWDM keyword enables the water relative permeability to be modified to account for formation damage. The KRWDM data consists of damage factors for each component tabulated against water relative permeability. The modified relative permeability values are then computed as

\[ K_{rw} = M_d K_{rw}' \]

where

\[ K_{rw}' \]

is the relative permeability prior to the damage calculation and

\[ M_d \]

is the damage multiplier given by

\[ M_d = \prod_i \frac{1}{1 + D_i(K_{rw}' \times \max(0, Z_i/I_i))} \]

where the product is taken over all hydrocarbon and water components specified with the COMPS and COMPW keywords:

\[ D_i \]

is the damage factor for a given \( K_{rw}' \) and component, found by interpolating the KRWDM data

\[ Z_i \]

is the molar concentration of component \( i \) (moles per reservoir volume).

The component molar concentrations can be output in the SUMMARY section using the BMLSC and BMWAT keywords, and in the SOLUTION and SCHEDULE section using the MLSC and MWAT mnemonics in the RPTRST, RPTSOL and RPTSCHED keywords.

\[ I_i \]

is the initial concentration at which damage occurs for a given component

See the KRWDI keyword.

The damage multiplier \( M_d \) can be written to the restart files by including the KRWDM item in the RPTRST keyword

The data comprises NTSFUN tables (see item 1 of keyword TABDIMS) each terminated by a slash (/). The tables must have at least 2 but not more than NSSFUN rows of data (see item 3 of keyword TABDIMS).

Each table consists of N columns of data, where N = 1 + number of hydrocarbon components + number of water components:

<table>
<thead>
<tr>
<th>Column</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Column 1</td>
<td>Water relative permeability</td>
</tr>
<tr>
<td></td>
<td>Values should be between 0 and 1 and should increase monotonically down the column.</td>
</tr>
<tr>
<td>Columns 2 to COMPS + 1</td>
<td>Damage factors for hydrocarbon components</td>
</tr>
<tr>
<td></td>
<td>UNITS: (kg-M/m³)⁻¹ (METRIC), (lb-M/rb)⁻¹ (FIELD), (gm-M/rcc)⁻¹ (LAB)</td>
</tr>
<tr>
<td>Columns COMPS + 2 to COMPS + COMPW + 1</td>
<td>Damage factors for water components</td>
</tr>
<tr>
<td></td>
<td>UNITS: (kg-M/m³)⁻¹ (METRIC), (lb-M/rb)⁻¹ (FIELD), (gm-M/rcc)⁻¹ (LAB)</td>
</tr>
</tbody>
</table>
Note: The damage factors must have values of zero or greater.

Note: The KRWDM keyword is only allowed in compositional simulations.

Example

With NTSFUN=2 and NSSFUN≥4, 6 hydrocarbon components and 1 water component. Damage occurring due to the presence of the sixth hydrocarbon component.

```
KRWDM
  0.0 5*0.0 0.9  0.0
  0.5 5*0.0 0.8  0.0
  0.9 5*0.0 0.75 0.0
  1.0 5*0.0 0.7  0.0
/
  0.0 5*0.0 0.8  0.0
  0.4 5*0.0 0.7  0.0
  0.7 5*0.0 0.6  0.0
  1.0 5*0.0 0.6  0.0
```
The KRW, KRWR, IKRW and IKRWR keywords scale the relative permeability of water within each grid block. KRW scales the Kr at the maximum water saturation (typically at a water saturation of 1.0), and KRWR scales the relative permeability at residual oil saturation (or residual gas in a gas-water run). The keyword can only be used when the end-point scaling option is active (keyword ENDScale in the RUNSPEC section).

If the KRWR keyword has not been used, the KRW keyword has the effect of scaling the relative permeability value calculated from the appropriate saturation table after the scaled saturation end-points have been accounted for. Hence:

\[ K_{rw} = K_{rw} \left( \text{Table} \right) \frac{KRW \left( \text{GridBlock} \right)}{K_{rwmax} \left( \text{Table} \right)} \]  
\[ \text{Eq. 3.103} \]

The relative permeability is taken to be the value at either the maximum saturation of the saturation table or at SWU if this has been specified.

If the KRWR keyword has been used, then the scaling honors the Kr at the critical saturation (SR) of the displacing phase. When the alternative scaling method using three saturation nodes is invoked by keyword SCALECRS, SR is given by

\[ \text{SR} = 1 - S_{OWCR} - S_{GL} \text{ in gas/oil or gas/oil/water runs} \]
\[ \text{SR} = 1 - S_{GCR} \text{ in gas/water runs} \]

where SOWCR and SGCN are the scaled values. When the alternative scaling method is not employed, only the two nodes SWCR and SWU are used to scale the saturation end-points, and SR is taken to be the transformed value of \( SR \) (Table) = 1.0 - \( S_{owcr} \) (Table) - \( S_{gcw} \) (Table) or \( SR \) (Table) = 1.0 - \( S_{gcw} \) (Table).

Hence the two cases are:

\[ SW < SR \quad K_{rw} = K_{rw} \left( \text{Table} \right) \frac{KRW \left( \text{GridBlock} \right)}{K_{rw} \left( \text{SR} \right)} \left( \text{Table} \right) \]  
\[ \text{Eq. 3.104} \]

\[ SW > SR \quad K_{rw} = KRW + \frac{KRW-KRWR}{K_{rwmax} - K_{rw} \left( \text{SR} \right)} \left( K_{rw} \left( \text{Table} \right) - K_{rw} \left( \text{SR} \right) \right) \]  
\[ \text{Eq. 3.105} \]

Fourteen keywords are provided to allow the maximum water relative permeability to be set for each grid cell independently. These keywords are:

\( \text{KRW} \quad \text{Max water } K_r \text{ for the non-directional saturation function tables} \)
\( \text{KRWX} \quad \text{Max water } K_r \text{ for the } +X \text{ face of each grid cell} \)
\( \text{KRWX-} \quad \text{Max water } K_r \text{ for the } -X \text{ face of each grid cell} \)
\( \text{KRWY} \quad \text{Max water } K_r \text{ for the } +Y \text{ face of each grid cell} \)
\( \text{KRWY-} \quad \text{Max water } K_r \text{ for the } -Y \text{ face of each grid cell} \)
\( \text{KRWZ} \quad \text{Max water } K_r \text{ for the } +Z \text{ face of each grid cell} \)
\( \text{KRWZ-} \quad \text{Max water } K_r \text{ for the } -Z \text{ face of each grid cell} \)
KRWR  $K_r$ at residual for the non-directional saturation function tables

KRWRX  $K_r$ at residual for the +X face of each grid cell

KRWRX-  $K_r$ at residual for the -X face of each grid cell

KRWRY  $K_r$ at residual for the +Y face of each grid cell

KRWRY-  $K_r$ at residual for the -Y face of each grid cell

KRWRZ  $K_r$ at residual for the +Z face of each grid cell

KRWRZ-  $K_r$ at residual for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the water $K_r$ value for flows in the +R,-R,+T,-T,+Z,-Z directions respectively.

None of the $KRW$ keywords should be used if water is not present in the model. The keyword ENDSCALE should also be specified in the RUNSPEC section.

If both directional and irreversible end-point scaling is requested using switches 'DIRECT' and 'IRREVERS' in keyword ENDSCALE, then the maximum water $K_r$ values for flow across the +X and -X faces of a grid cell are set using the keywords KRWX and KRWX-. Similarly KRWY and KRWY- are used for the +Y,-Y faces and the keywords KRWZ and KRWZ- for the +Z,-Z faces.

When the end-point scaling is directional but reversible ('DIRECT' but not 'IRREVERS' is specified in ENDSCALE), the keyword KRWX defines the maximum water $K_r$ values for both the +X and -X faces of each grid cell. Keywords KRWY, KRWZ are used for both the +Y,-Y faces and the +Z,-Z faces of each grid cell respectively.

If ENDSCALE is specified without either switch 'DIRECT' or 'IRREVERS', then none of the directional forms of the $KRW$ keyword are allowed. The $KRW$ keyword then defines the maximum water $K_r$ value for flow across each grid cell face.

Each $KRW$ keyword should be followed by one real number for each grid block in the current input box, specifying the maximum water relative permeability value. The data must be terminated by a slash(/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If any of the $KRW*$ keywords are omitted in a model where they could be accepted, then the maximum water $K_r$ value defaults to the value used in the appropriate water saturation function table.

If pure normalized curves are used where (1.0-SOWCR-SGL) is equals to SWU, then depending on the Scaling option used (see the SCALECRS keyword) the two end-points may be equal. If this is the case both KRW and KRWR apply to the same point, and the KRWR keyword is ignored.

If the Hysteresis model is active (see keyword SATOPTS), then the KRW, KRWR data scales the drainage curve. The imbibition curve can be scaled by using the equivalent IKRW (IKRWR, IKRWX, IKRWX-, ...) keyword.

If in addition to conventional hysteresis, WAG hysteresis has been specified (see the WAGHYSTR keyword) the additional end-point scaling keywords KRWH, KRWRH, IKRWH and IKRWRH data are available to scale the two-phase and three-phase WAG hysteresis water relative permeability saturations functions. These are
specified via the **WH2NUM** and **WH3NUM** keywords for saturation regions with WAG hysteresis applied to the water phase. The hysteresis models are described in more detail in "Hysteresis in WAG floods" in the ECLIPSE Technical Description.

See also keywords **SWU** and **ENKRVD** in the PROPS section.

Refer to "Saturation table scaling" in the ECLIPSE Technical Description for more information.

**Example**

```
-- In the RUNSPEC section :
DIMENS
  1  7  3 /
ENDSCALE
/
-- In the PROPS section :
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX : 1 1 2 7 1 3
KRW
  6*0.55
  6*0.65
  6*0.67 /
```
Use K-values in field simulation

This keyword requests that K-values be used to control liquid-vapor phase equilibrium. The actual K-values are entered using the **KVTABLE** keyword in the **PROPS** section.

The keyword contains no items.
Oil component K-value correlation

This keyword can only be used with the THERMAL option.

KVAN specifies up to three constants for each oil component. These constants are used to calculate an oil component K-value as a function of temperature and pressure. For each oil component, the K-value is fitted according to the equation

\[ K(P, T) = e^{A - \frac{B}{(T+C)/P}} \]

where

- \( T \) is the temperature (K (METRIC), \( \times \)R (FIELD), K (LAB), K (PVT-M)), and
- \( P \) is the pressure (barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)).

The keyword is followed by up to three rows, each containing \( N_c \) columns of data described below. The set of records is ended by a blank record, containing only a slash.

1. A, \( N_c \) values
   - UNITS: ln (P)
2. B, \( N_c \) values
   - UNITS: K (METRIC), \( \times \)R (FIELD), K (LAB), K (PVT-M)
3. C, \( N_c \) values
   - UNITS: K (METRIC), \( \times \)R (FIELD), K (LAB), K (PVT-M)

**Note:** This keyword is equivalent to KVCR. Internally ECLIPSE 300 will convert the coefficients of this keyword into KVCR coefficients and any reporting in the print file will be under the KVCR keyword.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

**Example**

```
KVAN
   -- C1 C2 HEAVY
   11.34 14.85 0
   1693 10252 0
   -7.16 -85.4 0
/
```
### KVCOMP

**K-value interpolation component**

KVCOMP specifies the component for which the K-value tables are interpolated. The K-value tables are specified with the `KVTABCn` keywords and the corresponding interpolation points using the `KVMP` keyword.

#### Example

```
KVCOMP 2 /
```
**KVCR**

**Oil component K-value correlation**

This keyword can only be used with the **THERMAL** option.

KVCR specifies up to five constants for each oil component. These constants are used to calculate an oil component K-value versus temperature and pressure. For each oil component, the K-value is fitted according to the equation

\[
K(P, T) = (A + B / P + C \cdot P) \cdot e^{-D/(T-E)}
\]

where

- \( T \) is the temperature (K (METRIC), °R (FIELD), K (LAB), K (PVT-M)), and
- \( P \) is the pressure (barsa (METRIC), psia (FIELD), atma (LAB), atm (PVT-M)).

**DEFAULT:** 0.0

**Note:** The K-value correlation only applies for temperatures greater than \( E \).

The keyword is followed by up to five rows, each containing \( N_c \) columns of data described below. The set of records is ended by a blank record, containing only a slash.

1. **A, \( N_c \) values**  
   **UNITS:** dimensionless
2. **B, \( N_c \) values**  
   **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
3. **C, \( N_c \) values**  
   **UNITS:** 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)
4. **D, \( N_c \) values**  
   **UNITS:** K (METRIC), °R (FIELD), K (LAB), K (PVT-M)
5. **E, \( N_c \) values**  
   **UNITS:** K (METRIC), °R (FIELD), K (LAB), K (PVT-M)

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of **TABDIMS**. The use of multiple EoS regions is discussed in **"Equations of state regions"** in the **ECLIPSE Technical Description**.

**Example**

```
KVCR
   -- HEAVY   SGAS
     212    1.23E6
     155.4E3 835.4E6
     0       0
     4000    16000
     480     0
/```

**Keywords**

1211
**KVCRS  Oil component K-value fitting function for surface conditions**

This optional keyword can be used to supply K-values for use in the surface calculations in THERMAL simulations. It should only be used when the KVCR keyword is used to set the reservoir K-values. If this keyword is not used, the surface K-values are set equal to the reservoir K-values.

This keyword is used in the same way as the KVCR keyword.

**DEFAULT: 0.0**

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.
**KVCRWAT**  

**Water K-value correlation**

This keyword can only be used with the **THERMAL** option.

KVCRWAT specifies up to five constants used to calculate the water K-value versus temperature and pressure. The K-value is evaluated according to the equation

\[
K(P, T) = (A + B / P + C \cdot P) \cdot e^{-D(T-E)}
\]

where

- \( T \) is the temperature (K (METRIC), °R (FIELD), K (LAB), K (PVT-M)), and
- \( P \) is the pressure (barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)).

**Note:** The current implementation requires the coefficients \( A \) and \( C \) to be zero and the coefficient \( B \) to be greater than zero.

The keyword is followed by up to five rows of data described below. The set of records is ended by a blank record, containing only a slash.

1. **A**
   - **UNITS:** dimensionless
   - **DEFAULT:** 0.0

2. **B**
   - **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
   - **DEFAULT:** None. Value must be greater than 0.0.

3. **C**
   - **UNITS:** 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)
   - **DEFAULT:** 0.0

4. **D**
   - **UNITS:** K (METRIC), °R (FIELD), K (LAB), K (PVT-M)
   - **DEFAULT:** 0.0

5. **E**
   - **UNITS:** K (METRIC), °R (FIELD), K (LAB), K (PVT-M)
   - **DEFAULT:** 0.0

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of **TABDIMS**. The use of multiple EoS regions is discussed in "Equations of state regions" in the **ECLIPSE Technical Description**.

**Example**

```
KVCRWAT
0
```
1.2E5
0
3800
46
/
KVGEN  Specify that K-values are to be generated internally

This keyword requests that a K-value table be generated internally. It is followed for each pressure table (from 1 to NTPVT as specified by item 2 of keyword TABDIMS) by a sample composition and the minimum and maximum pressures. The table will be K-value table 1.

Column 1 to $N_c$  Composition.

Column $N_c+1$  Minimum pressure.

UNITs: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

DEFAULT: 1 atma

Column $N_c+2$  Maximum pressure.

UNITs: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

DEFAULT: 500 atma

The KVALUES option

If KVGEN is used to generate a K-value table, the KVALUES keyword may be used in the RUNSPEC section to apply it to the simulation, or it may be used in a separator flash (SEPCOND or FIELDSEP).

The THERMAL option

In THERMAL mode, when an EoS (equation of state) environment is set by including CGDTYPE, CODTYPE, CGVTYPE or COVTYPE, the K-value tables $KVTABT_1$ ... $KVTABT_9$ ($KVTABT_n$) are filled using the EoS. Temperatures for the tables need to be set using KVTEMP. Note that only the first EoS region number is used when generating the tables. For pressures below the dew point or above the bubble point, the last two-phase K-value is expanded as $K_{bub} P_{bub} / P$ and $K_{dew} P_{dew} / P$.

Example

Generate K-values for the specified sample from 10 to 2500 psia.

```
KVGEN 0.0247 0.2516 0.2407 0.0733 0.4097 10 2500 /
```
KVMF specifies the mole fractions of the nominated component at which K-value tables are available. The interpolation component is specified using the KVCOMP keyword.

The number of K-value tables and hence the number of values required by this keyword is set with the 24th item in the TABDIMS keyword in the RUNSPEC section. Values must lie in the interval [0,1] and be monotonically increasing.

**Example**

```
KVMF 0.0 0.5 1.0 /
```
The keyword family KVTABCn (where n is an integer between 1 and 9) enables K-values to be entered to control liquid-vapor phase equilibrium at differing compositions. The compositions are specified as a mole fraction of a nominated component using the KVMP and KVCOMP keywords.

KVTABCn is a pressure table, and it is possible to enter more than one table; these can be assigned to cells using the PVTNUM keyword in the REGIONS section.

The table has \( N_c + 1 \) columns, where \( N_c \) is the number of components in the run. The first column is of pressure values, the remaining columns are the K-values for each component at these pressures. The K-values should be positive and greater than \( 10^{-6} \).

**Example**

\[
\begin{array}{lll}
\text{KVTABC1} & \text{5} & 0.1197 \ 0.0192 \\
& 50 & 0.0128 \ 0.0020 \\
& 95 & 0.0071 \ 0.00011 \\
& 140 & 0.0051 \ 0.00081 \\
/ & & \\
\end{array}
\]
KVTABLE

Tables of K-values

This keyword enables K-values to be entered to control liquid-vapor phase equilibrium. If the KVALUES keyword was specified in the RUNSPEC section, these are used instead of the equation of state equal fugacity relationships to obtain the composition of the liquid and vapor hydrocarbon phases in a cell, and the split of the total moles in the cell between these phases.

The K-values may also be used to control separators.

KVTABLE is a pressure table. It is possible to enter more than one table, and these can be assigned to cells using the PVTNUM keyword in the REGIONS section.

The table has \( N_c + 1 \) columns, where \( N_c \) is the number of components in the run. The first column is of pressure values, the remainder the K-values for each component at these pressures.

Column 1

The oil phase pressure.

Values should increase monotonically.

**UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

Column 2 to \( N_c + 1 \)

The corresponding K-values for each component.

The K-value for component \( c \) is \( K_c = y_c / x_c \), where \( y_c \) and \( x_c \) are mole fractions of component \( c \) in vapor and liquid phases respectively.

**Example**

```
 3014.7  1.14  1.44  1.01  0.92  0.73  0.12  0.041
 5014.7  1.66  1.54  1.12  0.85  0.60  0.09  0.023
/```

Keywords
Tables of K-values at T=Tn (n = 1 to 9)

The keyword family KVTABTn (where n is an integer between 1 and 9) enables K-values to be entered to control liquid-vapor phase equilibrium at the temperature Tn, corresponding to the nth value specified in the KVTEMP keyword.

These keywords can only be used with the THERMAL option.

KVTABTn is a pressure table, and it is possible to enter more than one table; these can be assigned to cells using the PVTNUM keyword in the Regions section.

The table has \( N_c + 1 \) columns, where \( N_c \) is the number of components in the run. The first column is of pressure values, the remaining columns are the K-values for each component at these pressures. The K-values should be positive and greater than \( 10^{-6} \).

Column 1  The oil phase pressure.

Values should increase monotonically.

UNITs: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

Columns 2 to \( N_c + 1 \)  The corresponding K-values for each component. All values should be greater than zero.

The K-value for component \( c \) is \( K_c = \frac{y_c}{x_c} \), where \( y_c \) and \( x_c \) are mole fractions of component \( c \) in vapor and liquid phases respectively.

If temperatures are encountered that are less than T1, then the K-value is linearly extrapolated to 0 at an absolute temperature of 0.

Note: Thermal simulation stability can be affected by the pressure spacing with this keyword. It is recommended that equal intervals are used between the pressures.

Note: There may be convergence problems in the thermal flash if all the K-values in a row are 1.

See also keyword KVTEMP.

Example

```
KVTAB1
  5  .3197  .0192
 50  .0128  .0020
 95  .0071  .0011
140  .0051  .00081
```
KVTEMP

K-value temperatures

This keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section).

KVTEMP specifies the temperatures at which K-value tables are available.

The number of temperatures equals the number set in item 15 in the TABDIMS keyword in the RUNSPEC section.

UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

Note: Thermal simulation stability can be strongly affected by the temperature spacing with this keyword. It is recommended that equally spaced intervals are used.

See also keyword KVTABTn.

Example

```
KVTEMP
  300 400 500 600
/
```
**Oil component K-value fitting function**

This keyword can only be used with the **THERMAL** option.

For each oil component, the oil component K-value is fitted according to the equation

\[
K(P, T) = e^{5.372697 \cdot (1+A_c) \cdot (1-T_{crit}/T)} \cdot P_{crit}/P
\]

where

- \(T\) is the temperature (K (METRIC), °R (FIELD), K (LAB), K (PVT-M))
- \(P\) is the pressure (barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M))
- \(A_c\) is the acentric factor for each component as entered with keyword **ACF**
- \(T_{crit}\) is the component critical temperature as entered with keyword **TCRIT**
- \(P_{crit}\) is the component critical pressure as entered with keyword **PCRIT**.

This keyword requires acentric factors with keyword **ACF**, critical temperatures and pressures with keywords **TCRIT** and **PCRIT**.

There is no data for this keyword.

**Example**

```
KVWI
```

---

**Keywords**

1221
Tables of Kw-values at \( T=T_n \) (\( n = 1 \) to 9)

The keyword family \texttt{KWTABTn} (where \( n \) is an integer between 1 and 9) enables K-values to be entered to control water-vapor phase equilibrium at the temperature \( T_n \), corresponding to the \( n \)th value specified in the \texttt{KWTEMP} keyword.

These keywords can only be used with the \texttt{THERMAL} option.

\texttt{KWTABTn} is a pressure table, and it is possible to enter more than one table; these can be assigned to cells using the \texttt{PVTNUM} keyword in the \texttt{REGIONS} section.

The table has \( N_c + 1 \) columns, where \( N_c \) is the number of components in the run. The first column is of pressure values, the remaining columns are the K-values for each component at these pressures. The K-values should be positive and greater than \( 10^{-6} \).

**Column 1**

- The pressure.
- Values should increase monotonically.
- UNITS: barsa (\texttt{METRIC}), psia (\texttt{FIELD}), atma (\texttt{LAB}), atma (\texttt{PVT-M})

**Columns 2 to \( N_c + 1 \)**

- The corresponding K-values for each component.
- The K-value for component \( c \) is \( K_c = y_c / a_c \), where \( y_c \) and \( a_c \) are mole fractions of component \( c \) in the vapor and aqua phases respectively.

If temperatures are encountered that are less than \( T_1 \), then the K-value is linearly extrapolated to 0 at an absolute temperature of 0.

**Note:** Thermal simulation stability can be affected by the pressure spacing with this keyword. It is recommended that equal intervals are used between the pressures.

**Note:** In a thermal \texttt{CO2STORE} run, the \( \text{CO}_2/\text{H}_2\text{O} \) component can be defaulted.

See also keyword \texttt{KWTEMP}.

**Example**

A two component system

```plaintext
KWTABT1
  5  .1197  .0192
  50  .0128  .0020
  95  .0071  .0011
 140  .0051  .00081
```
**KWTEMP**

**Kw-value temperatures**

This keyword can only be used with the **THERMAL** option (keyword **THERMAL** in the **RUNSPEC** section).

**KWTEMP** specifies the temperatures at which Kw-value tables are available.

The number of temperatures equals the number set in item 15 in the **TABDIMS** keyword in the **RUNSPEC** section, with a maximum of 9 values.

**UNITS:** °C (**METRIC**), °F (**FIELD**), °C (**LAB**), °C (**PVT-M**)

---

**Note:** Thermal simulation stability can be strongly affected by the temperature spacing with this keyword. It is recommended that equally spaced intervals are used.

See also keyword **KWTABTn**.

**Example**

The temperature for **KWTABT1**, **KWTABT2**, **KWTABT3** and **KWTABT4**.

```
KWTEMP
  300 400 500 600
```
This section contains an alphabetic listing of keywords starting with the letter L.
Lab units are to be used

This indicates that lab units are to be used. Note that the default unit convention is METRIC.

See "Units" in the ECLIPSE Technical Description for further details.

The keyword has no associated data.
# LAGTRAC

## Activates Lagrangian tracer solver option

This keyword activates the Lagrangian tracer solver option. For more information on this option see "Lagrangian tracer solver" in the *ECLIPSE Technical Description*.

The data consists of four items, describing the maximum number of streamlines and streamline segments, along with the target number of streamlines and the minimum number of streamlines for each cell. The data must be terminated by a slash (/).

1. The maximum number of streamlines.
   
   **DEFAULT:** \((\text{Number of components} + \text{number of phases}) \times \text{number of cells in model} \times 0.1\).

2. The maximum number of streamline segments. Each streamline is made up of a number of segments, with the number of segments being one greater than the number of cells the streamline passes through. This entry is the maximum number of streamline segments which can be traced in total, that is the sum of the number of segments for all streamlines.
   
   **DEFAULT:** \(100 \times \text{maximum number of streamlines}\).

3. The target number of streamlines as a fraction of the total number of cells in the grid.
   
   **DEFAULT:** 0.1

4. The minimum number of streamlines to pass through any cell.
   
   **DEFAULT:** 1

### Note:

- The Lagrangian tracer solver is not compatible with the radial local grid refinement.
- The Lagrangian tracer solver is not compatible with local grid refinements where two refined regions are immediately adjacent, or a refinement is immediately adjacent to a fault. A fault running through a refined area is also not permitted.
- The Lagrangian tracer solver is not compatible with dual porosity.
- The Lagrangian tracer solver is not compatible with explicit NNCs set using **NNC** keyword.

### Example

Up to 4000 streamlines comprising up to 100000 streamline segments. The algorithm will initially trace approximately 0.2 streamlines per cell, and will then add streamlines until every cell contains at least one streamline segment.

```
LAGTRAC 4000 100000 0.2 1 /
```
LAGTRUPD  Specifies the update criteria for the Lagrangian tracer solver option

The data consists of four items, describing the maximum time between retraces of the streamlines, the maximum time between updates of tracer concentrations of the streamlines, the maximum change in any completion flow rate before retraces of the streamlines, and the tracer solver method. The data must be terminated by a slash (/).

1. The maximum time between retraces of the streamlines.
   
   **UNITS:** day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
   
   **DEFAULT:** 0.0

2. The maximum time between updates of the tracer concentration on the streamlines.

   **UNITS:** day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
   
   **DEFAULT:** 0.0

3. The maximum fractional change in completion component flow rate before a streamline retrace is forced.

   **DEFAULT:** 0.05

4. Tracer solver method:

   **ADVEC**  Tracer concentrations are advected along streamlines and averaged at each cell.

   **COMPR**  as ADVEC with compression taken into account.

   **DEFAULT:** COMPR

**Example**

Streamlines will be retraced every 182.5 days or when any completion changes its flow rate by 10% in any component. Tracer concentrations are stored on streamlines at trace, and then every 60 days thereafter. Cell tracer concentrations are derived by advecting tracer concentrations along streamlines from the last time when they were written to streamlines, and then averaging over streamlines in each cell.

```
LAGTRUPD 182.5 60.0 0.1 'ADVEC' /
```
**Extended Langmuir Isotherm**

This keyword is specific to the compositional coal bed methane option.

The data determines the Langmuir sorption data for each hydrocarbon component in the run. A coal bed methane model can be divided into one or more regions. The number of regions is `NTCREG` and this number is input by `REGDIMS` item 6. Coal regions are input with the keyword `COALNUM`, while coal densities are input using the keyword `ROCKDEN`. There is one `LANGMEXT` record for each coal region up to `NTCREG` records. For further information see "Coal bed methane model" in the *ECLIPSE Technical Description*.

Each record contains the Langmuir pressure constants $P_i$ and the Langmuir volume constants $V_i$ for all components, totaling $2 \times NCOMPS$ entries:

Record 1

$$P_1, V_1, P_2, V_2, \ldots, P_{ncomps}, V_{ncomps}$$

- $P_i$ has the units of pressure. Values should be larger than zero. The exception is if the volume constant is zero, then the component is completely ignored when calculating the adsorption capacities.
  
  UNITS: bars (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)
  
  DEFAULT: 0.0

- $V_i$ is given as surface volume of gas over coal weight. The values should be larger or equal to zero.
  
  UNITS: sm$^3$/kg (METRIC), Mscf/ton (FIELD), scc/gm (LAB), sm$^3$/kg (PVT-M)
  
  DEFAULT: 0.0

Record 2

$$P_1, V_1, P_2, V_2, \ldots, P_{ncomps}, V_{ncomps}$$

Defines the Langmuir constants for coal region number two.

Records 3 to `NTCREG`:

Each record defines the Langmuir constants for coal regions 3 to `NTCREG`.

A record is terminated by a slash (/) a second slash terminates the keyword.

The number of records should equal the maximum number of coal regions as set with keyword `COALNUM`.

Each pressure constant $P_i$ and volume constant $V_i$ defines a Langmuir curve for component $i$. The volume constants define the maximum storage capacity and are input as surface volume of gas over coal-weight. The pressure constant defines the pressure at half of the Langmuir volume.

**Note:** In FIELD units the storage constant is not given in the natural units of ECLIPSE but instead as Mscf/ton (where 1 ton = 2000 lb; the so-called short ton). If item 112 in `OPTIONS3` is set to a non-zero value, the ton is taken to be a long ton (2240 lb).

**Example**

Three components and two coal regions:
<table>
<thead>
<tr>
<th>Pressure (psi)</th>
<th>Mscf/ton</th>
<th>Component</th>
</tr>
</thead>
<tbody>
<tr>
<td>276.0</td>
<td>0.9938</td>
<td>CO₂</td>
</tr>
<tr>
<td>3951.0</td>
<td>0.482</td>
<td>N₂</td>
</tr>
<tr>
<td>680.0</td>
<td>0.486/</td>
<td>CH₄</td>
</tr>
<tr>
<td>276.0</td>
<td>0.961</td>
<td>CO₂</td>
</tr>
<tr>
<td>3951.0</td>
<td>0.441</td>
<td>N₂</td>
</tr>
<tr>
<td>680.0</td>
<td>0.442/</td>
<td>CH₄</td>
</tr>
</tbody>
</table>
**LANGMPL**

### Langmuir pressure scaling

This keyword may be used with the coal bed methane model (see "Coal bed methane model" in the *ECLIPSE Technical Description*).

The keyword scales the pressure used when calculating the adsorption capacity on a cell by cell basis.

The keyword should be followed by one real number for every grid block, specifying a multiplier. The data field should be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword `DIMENS`. Repeat counts may be used for repeated values (for example 115*0.9). Note that spaces must not be inserted on either side of the asterisk (*).

Only the values given for the coal cells are used. See the `COALNUM` keyword. So for a dual porosity run with only the matrix cells being coal cells, it is sufficient to supply NDIVIX * NDIVIY * NDIVIZ / 2 values. In the case of the ECLIPSE 300 instant sorption model values must be applied for all cells with coal.

The keyword is entirely optional and if not specified, or defaulted, a value of unity will be used.

**Example**

With NDIVIX = 10, NDIVIY = 1 and NDIVIZ = 2 specified in RUNSPEC keyword `DIMENS`:

```
LANGMPL
10*0.8 /
```

The pressures of the first 10 grid cells (i.e. the coal matrix cells) are multiplied with 0.8 when computing the adsorption capacity.

**Example - ECLIPSE 300 instant sorption**

With NDIVIX=10, NDIVIY=1 and NDIVIZ=2 specified in RUNSPEC keyword `DIMENS`:

```
LANGMPL
20*0.8 /
```

The pressures of the first 20 grid cells are multiplied with 0.8 when computing the adsorption capacity.
LANGMUIR  Tables of coal surface gas concentration

This keyword is specific to the coal bed methane option.

The data comprises NTCREG (see keyword REGDIMS, item 6) tables of Langmuir isotherm data, each terminated by a slash (/). For ECLIPSE 100, each table consists of 2 columns of data, while for ECLIPSE 300 each table consists of NCOMPS + 1 columns of data.

1  The fracture pressure.
   In ECLIPSE 300, the partial pressure, \( y_i P \), is used to look up the adsorption capacity for each component, where \( y_i \) is the gas phase mole fraction of component \( i \) and \( P \) is the connected pore volume/fracture pressure.

   The pressure used to look up the adsorption capacity is scaled with the coal grid cell LANGMPL value if this keyword is present.

   Values should increase monotonically down the column.

   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atm (PVT-M)

2  The corresponding surface gas concentration (the first component in ECLIPSE 300)
   Values should be increase monotonically down the column.

   UNITS: sm\(^3\)/m\(^3\) (METRIC), Mscf/ft\(^3\) (FIELD), scc/cc (LAB)

3 to NCOMPS + 1 (ECLIPSE 300 only)
   The corresponding surface gas concentration for each of the remaining components.

   Values should be level or increasing down the column.

   UNITS: sm\(^3\)/m\(^3\) (METRIC), Mscf/ft\(^3\) (FIELD), scc/cc (LAB)

The entire table may be defaulted, provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

The coal region number COALNUM is used to look up the table.

There must be the same number of entries in each column of a given table. This number (rows) should not be less than 2 or greater than NSSFUN (item 3 in keyword TABDIMS).

If OPTIONS keyword item 180 is activated, the number of tables should equal NTSFUN (see keyword TABDIMS) and the region keyword SATNUM is used to look up the table.

See also the LANGSOLV and COALADS keywords and "Coal bed methane model" in the ECLIPSE Technical Description for further information.

Examples

**ECLIPSE 100 Example**

With NTCREG=2 and NSSFUN\(\geq7\)

<table>
<thead>
<tr>
<th>LANGMUIR</th>
<th>0.0 0.0</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1000.0 0.050</td>
</tr>
<tr>
<td></td>
<td>2000.0 0.065</td>
</tr>
<tr>
<td></td>
<td>3000.0 0.083</td>
</tr>
</tbody>
</table>
### ECLIPSE 300 Example

With NCOMPS=2, NTCREG=1 and NSSFUN≥20

```plaintext
LANGMUIR
   --p       CO2            CH4
   --bars     SM3/M3        SM3/M3
0        0              0          -- first entry should be zero
1     1.72407349        0.35559202
10   11.89523500        2.97648839
20   17.69503828        5.06294513
30   21.12902337        6.60665445
40   23.39953732        7.79501835
50   25.01221730        8.73806793
100  29.01106209        11.52721805
110  29.43893197        11.87170836
120  29.80525113        12.17491392
150  30.64414648        12.89972896
200  31.53163238        13.71630860
300  32.47205642        14.64325719
400  32.96362284        15.15535696
500  33.26577192        15.48017795
600  33.47030083        15.70457275
700  33.61793950        15.86879375
800  33.72952610        15.99438341
900  33.81682914        16.09337862
1000 33.88699764        16.17346150
```

---

**Keywords**

1232
**LANGMULC**  Langmuir multiplier for each component

This keyword may be used with the coal bed methane model (see "Coal bed methane model" in the *ECLIPSE Technical Description*).

The keyword scales the adsorption capacity on a cell by cell basis for each component. The keyword should be followed by one real number for every grid block, specifying a multiplier for the adsorption capacity. The data field should be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest (from 1 to \texttt{NDIVIX}) followed by the Y axis index (from 1 to \texttt{NDIVIY}) and then the Z axis index (from 1 to \texttt{NDIVIZ}) for each component. \texttt{NDIVIX}, \texttt{NDIVIY} and \texttt{NDIVIZ} are specified in \texttt{RUNSPEC} keyword \texttt{DIMENS}. Repeat counts may be used for repeated values (for example 115*0.9). Note that spaces must not be inserted on either side of the asterisk (*).

Only the values given for the coal cells are used. See \texttt{COALNUM}.

The keyword is entirely optional and if not specified, or defaulted, the adsorption capacity as specified by \texttt{LANGMEXT} or \texttt{LANGMUIR} is used unaltered.

*Note:* This keyword cannot be used with \texttt{LANGMUL}, \texttt{GASSATC} or \texttt{GASADEC}.

**Example**

With \texttt{NDIVIX=10, NDIVIY=1} and \texttt{NDIVIZ=2} specified in \texttt{RUNSPEC} keyword \texttt{DIMENS}:

```plaintext
LANGMULC
  5* 5*0.2 10*0.0 -- component 1
  5* 5*0.4 10*0.0 -- component 2/
```

The first 5 cells are unaltered, while the 5 next cells are scaled to 20% of the adsorption capacity, for the first component, the second component is scaled with 40% of the adsorption capacity as given by the Langmuir isotherm.
LANGMULC  Langmuir multiplier

This keyword may be used with the coal bed methane model (see "Coal bed methane model" in the ECLIPSE Technical Description).

The keyword scales the adsorption capacity on a cell by cell basis. The keyword should be followed by one real number for every grid block, specifying a multiplier for the adsorption capacity. The data field should be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*0.9). Note that spaces must not be inserted on either side of the asterisk (*).

Only the values given for the coal cells are used, see COALNUM. So for a dual porosity run with all matrix cells being coal cells, it is sufficient to supply NDIVIX * NDIVIY * NDIVIZ /2 values.

The keyword is entirely optional and if not specified, or defaulted, the adsorption capacity as specified by LANGMEXT or LANGMUIR is used unaltered.

Note: This keyword cannot be used with LANGMULC, GASSATC or GASADEC.

Example

With NDIVIX=10, NDIVIY=1 and NDIVIZ=2 specified in RUNSPEC keyword DIMENS:

```
LANGMULT
5* 5*0.2 /
```

The first 5 cells are defaulted to unity, while the 5 next cells are scaled to 20% of the adsorption capacity as given by the Langmuir isotherm. The remaining cells, not specified, are set to the default value of unity.
**LANGSOLV  Tables of coal surface solvent concentration**

This keyword is specific to the coal bed methane option.

The data comprises NTCREG (see keyword REGDIMS, item 6) tables of Langmuir isotherm data for the solvent component, each terminated by a slash (/). Each table consists of 2 columns of data:

1. The fracture pressure.
   - Values should increase monotonically down the column.
   - The pressure used to look up the adsorption capacity is scaled with the coal grid cell LANGMPL values if this keyword is present.
   - UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2. The corresponding surface solvent concentration.
   - Values should be level or increasing down the column.
   - UNITS: sm$^3$/m$^3$ (METRIC), Mscf/ft$^3$ (FIELD), scc/cc (LAB)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (item 1 in keyword TABDIMS).

The entire table may be defaulted, provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

The coal region number COALNUM is used to look up the table.

If item 180 in OPTIONS is activated, the number of tables should equal NTSFUN (see keyword TABDIMS) and the region keyword SATNUM is used to look up the table.

See also the LANGMUIR and COALADS keywords and "Coal bed methane model" in the ECLIPSE Technical Description for further information.

**Example**

With NTCREG=2 and NSSFUN≥7

```
LANGSOLV
  0.0  0.0
  1000.0  0.050
  2000.0  0.075
  3000.0  0.093
  4000.0  0.100
  6000.0  0.110
/
--
  0.0  0.050
  1000.0  0.060
  2000.0  0.075
  3000.0  0.093
  4000.0  0.100
  6000.0  0.110
```

**Keywords**

1235
LBCCOEF

Set non-default LBC coefficients

The keyword enables the default coefficients used by the Lorentz-Bray-Clark viscosity correlation to be modified. This should only be done with great care; the viscosity is obtained from a fourth order polynomial in reduced density and must clearly not go negative.

There are five coefficients with default values of 0.1023, 0.023364, 0.058533, -0.040758, 0.0093324. Any that are not specified with LBCCOEF take these default values.

Only one set of LBC coefficients is required.

The data must be terminated with a slash (/).

Example

Reset two of the LBC coefficients:

```
LBCCOEF
1* 0.025 1* 0.01 /
```
**LBCCOEFR**  **Set non-default LBC coefficients for multiple equation of state regions**

The keyword enables the default coefficients used by the Lorentz-Bray-Clark viscosity correlation to be modified for each equation of state region. This should only be done with great care; the viscosity is obtained from a fourth order polynomial in reduced density and must clearly not go negative. Equation of state regions are defined using the **EOSNUM** keyword.

There are five coefficients with default values of 0.1023, 0.023364, 0.058533, -0.040758, 0.0093324. Any that are not specified with **LBCCOEFR** take these default values.

A set of coefficients must be supplied for each equation of state region, with each set terminated with a slash (/).

**Example**

Set LBC coefficients in a model with two equation of state regions:

```
LBCCOEFR
  0.1 /
  1* 0.025 1* 0.01 /
```
Units for linear combination of phases

This keyword sets the units of the linear combination of oil, water and gas as defined in keyword LINCOM. This keyword is optional and used for reporting purposes only. If it is not entered or defaulted the default string is used.

For example, the volume of a quantity, \( V_s \), may be equal to

\[
V_s = \alpha V_{oil} + \beta V_{wat} + \gamma V_{gas}
\]

Eq. 3.107

and the flow rate of this quantity, \( Q_s \), equal to

\[
Q_s = \alpha Q_{oil} + \beta Q_{wat} + \gamma Q_{gas}
\]

Eq. 3.108

for coefficients \( \alpha, \beta, \gamma \). The units of \( V_s \) are specified with this keyword which determines the units of \( \alpha, \beta, \gamma \) (see LINCOM). The units of \( Q_s \) are assumed to be those of \( V_s \) per day (METRIC, FIELD, PVT-M) or per hour (LAB).

The keyword is followed by a single line of data terminated with a slash (/).

1. Up to 4 characters to define the units of \( V_s \).

DEFAULT: UNIT

Example

```
LCUNIT
SM3 /
```
LGR

Set options for LGR and coarsening

This keyword sets options and dimensions for local grid refinement and coarsening. (See “Local grid refinement and coarsening” in the ECLIPSE Technical Description for further information.)

The data consists of some or all of the following items, describing the dimensions and switches required for the local grid refinement and coarsening options. The data must be terminated by a slash (/).

1. **MAXLGR** The maximum number of LGRs in the model.
   
   **DEFAULT:** 0

2. **MAXCLS** The maximum number of cells in each LGR

   **DEFAULT:** 0

3. **MCOARS** The maximum number of amalgamated coarse cells

   **DEFAULT:** 0

4. **MAMALG** The maximum number of LGR amalgamations

   **DEFAULT:** 0

5. **MXLALG** The maximum number of LGRs in any amalgamation

   **DEFAULT:** 0

6. **LSTACK** The length of the stack of previous search directions used by the linear solver for LGR simulations. For cases where LGRs are defined, if LSTACK is zero or negative, it defaults to **NSTACK**, the global stack value.

   **DEFAULT:** **NSTACK** (10)

7. Specify pressure interpolation for LGR runs

   **INTERP** This specifies that pressures are to be bilinearly interpolated spatially in global cells neighboring a Cartesian local grid refinement to enhance the accuracy of flow calculations between local and global cells (sometimes known as the Quandalle correction). This option is ignored in VE runs or for radial local grids. This is performed as an integrated part of the fully implicit pressure solution.

   **NOINTERP** Pressure interpolation is not to be used

   **DEFAULT:** **NOINTERP**

8. **NCHCOR** The total number of grid cells contained in grid coarsenings that straddle domain boundaries in Parallel runs.

   **DEFAULT:** 0

Example

```
LGR --MAXLGR  MAXCLS  MCOARS  MAMALG  MXLALG  LSTACK  Pressure      NCHCOR
     --                                                Interpolation
2   400     0       1       2 /
```

Keywords

```
LGR
ECLIPSE 100
ECLIPSE 300
SPECIAL
RUNSPEC
GRID
EDIT
PROPS
REGIONS
SOLUTION
SUMMARY
SCHEDULE
```
LGRCOPY  Allow LGRs to automatically inherit rock property data from host cells

This keyword requests an option for processing the local grid refinement data. It allows the LGR rock properties to be automatically inherited from their global host cells when the LGR is defined, instead of at the end of processing for the GRID section. This will in turn allow the user to use an operator keyword on the LGR cells without the need to set LGR cells first.

If the LGRCOPY keyword is present in the RUNSPEC section, this option will apply to every LGR in the model. If individual LGRs request this option, then this keyword can be used inside the relevant CARFIN / ENDFIN, RADFIN / ENDFIN, REFINE / ENDFIN pair.

This keyword has no associated data.

Treatment without LGRCOPY

If LGRCOPY is not specified, values of porosity, permeability and so on, are copied to local grid cells from the global host cells at the end of processing the GRID section. In most cases this is adequate, but it is also possible to define properties directly for some, or all, of the cells within a local grid by specifying grid arrays between the CARFIN / ENDFIN, RADFIN, RADFIN4 / ENDFIN or REFINE / ENDFIN pairs.

When the end of the GRID section is reached, any values that have not been explicitly defined for local cells are then copied from the global host cells.

For example:

```
CARFIN
  'LGR1'  5 10   5 10   1 4 18 18  8   5  /
  EQUALS
  'PORO'  0.3  1 18 1 18 1 4 /
  'PERMZ' 50.0 1 18 1 18 1 4 /
  'NTG'   0.9  1 18 1 18 1 4/
  /
  MULTIPLY
  'PORO'  0.5  6 13 6 13 1 2 /
  'NTG'   0.2  6 13 6 13 1 2 /
  /
ENDFIN
```

This defines the porosities only in the top four layers on the local grid, and these are subsequently modified for the top two layers with the MULTIPLY keyword. The porosities for cells in the bottom four layers are copied from their host cells in the global grid.

Since properties are not copied from host cells until the end of the GRID section is reached, when LGRCOPY is not present it is not possible to modify any property for cells within the local grid (by using the ADD, COPY or MULTIPLY keywords, for instance) without previously defining the property for those cells within the LGR, as shown in the above example.

Treatment with LGRCOPY

When LGRCOPY is specified, values for any cells within the LGR which have not been explicitly defined will be copied from the global cells ‘just in time’ to allow property modification. It is important to note that the order of property specification in the dataset becomes relevant here. For instance, if the LGR uses non-default values for DX, these must be have been specified before the modification keyword if they are to be taken into account when determining which global cell each local cell lies in.
Consider the previous example without \texttt{LGRCOPY} use. Say, the model contains only one LGR, so the \texttt{LGRCOPY} keyword could either be specified in the \texttt{RUNSPEC} section to apply to that LGR, or placed just after \texttt{CARFIN} definition (as in this example). It is possible to have:

\begin{verbatim}
PORO 0.3 5 10 5 10 1 2 /
NTG 0.9 5 10 5 10 1 2 /
CARFIN 'LGR1' 5 10 5 10 1 4 18 18 8 5 /
LGRCOPY EQUALS 'PERMZ' 50.0 1 18 1 18 1 4 /
/
MULTIPLY 'PORO' 0.5 6 13 6 13 1 2 /
'NTG' 0.2 6 13 6 13 1 2 /
/
ENDFIN
\end{verbatim}

This defines \texttt{PORO} and \texttt{NTG} for the global cells which correspond to the top four layers on the local grid, so is the exact analogy of the example above. It would be more useful if, say, these values were also required for a larger range of global cells of which the LGR was a subset. Using \texttt{LGRCOPY}, the range of cells in the global specification just needs to be expanded, whereas without it the dataset needs two specifications for each keyword: one for the global cells and another one for the local cells.

Similarly, \texttt{LGRCOPY} will invoke automatic inheritance of global properties within a \texttt{REFINE/ENDFIN} pair in the \texttt{EDIT} section.

\textit{CAUTION:} The \texttt{CARFIN/RADFIN/REFINE...ENDFIN} keywords must be placed after the global properties in the dataset, in order for the local properties to be automatically inherited from the global properties.

\section*{Notes}
\begin{itemize}
  \item It is not possible to use the \texttt{LGRCOPY} keyword with keyword \texttt{DPGRID}.
  \item This keyword cannot be used with the \texttt{EXTFIN/ENDFIN} set for specifying unstructured local grids.
  \item The \texttt{LGRCOPY} keyword can be used with the keywords \texttt{EQUALS, ADD, MULTIPLY, COPY, MAXVALUE, MINVALUE} and \texttt{COPYBOX}.
\end{itemize}

\textbf{ECLIPSE 300 only}
\begin{itemize}
  \item For integer-valued keywords (\texttt{MULTNUM, OPERNUM} and \texttt{FLUXNUM}) in the \texttt{GRID} section only, if only a subset of the local cells have values set by the user inside an LGR, \texttt{LGRCOPY} is unable to copy global values to unset LGR cells.
  \item It is not possible to use the \texttt{LGRCOPY} keyword together with the \texttt{DX, DY, DZ, TOPS, MIDS, MULTX, MULTY, MULTZ, MULTX-, MULTY-} and \texttt{MULTZ-} keywords. If local values for these properties are required, they need to be set by the user within the refinement, or the properties will be copied from host cells when the end of the \texttt{GRID} section is reached.
\end{itemize}

\textbf{ECLIPSE 100 only}
\begin{itemize}
  \item Use of the \texttt{AMALGAM} keyword does not affect \texttt{LGRCOPY}. If used in the \texttt{GRID} section, \texttt{LGRCOPY} must still be specified for each individual LGR as required.
\end{itemize}
• For radial LGRs, only properties which also exist in the Cartesian global grid can be copied using LGRCOPY - for instance, PERMR cannot be copied since the global grid has PERMX and PERMY.

Example

```
LGRCOPY
```

Keywords

1242
Solves a local grid refinement using local timestepping

LGRFREE can be used at any time during the run to designate that a local grid should be solved using local timestepping. If a local grid has previously been designated as an in-place refinement using the LGRLOCK keyword, the LGRFREE keyword returns it to local timestepping (which is the default solution method for local grid refinements).

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Name of the local grid refinement - up to 8 characters.
   A local grid name root ending in an asterisk (*) may be used to refer to several local grids on the same record.

If the keyword is entered without any data records (that is followed only by a blank record), all local grids are solved using local timestepping.

If the local grid is part of an amalgamation specified using the keyword AMALGAM, the whole amalgamation is solved using local timestepping.

See also keyword LGRLOCK.

Example

This returns the 'in-place' refinement SOUTH to local timestepping.

```plaintext
LGRFREE
SOUTH / 
/ 
```
LGRLOCK **Solves a local grid refinement in-place**

LGRLOCK can be used at any time during the run to designate that a local grid should be solved in-place. That is, local timestepping is not used and the local grid is solved fully implicitly with the global grid.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Name of the local grid refinement - up to 8 characters.
   
   A local grid name root ending in an asterisk (*) may be used to refer to several local grids on the same record.

If the keyword is entered without any data records (that is followed only by a blank record), all local grids is run in-place.

If the local grid is part of an amalgamation specified using the keyword **AMALGAM**, the whole amalgamation is solved in-place.

See also keyword **LGRFREE**.

**Example**

This turns the refinement SOUTH and any refinements with root N* into 'in-place' refinements.

```
LGRLOCK
  'SOUTH' /
  'N*' /
/  
```

---

**Keywords**

1244
LGROFF can be used at any time during the run to switch a local grid refinement off. Optionally, the local grid refinement can be switched on and off automatically depending on the number of active wells it contains. (A well is active if it is open or stopped, but not shut.)

The keyword is followed by either one or two items of data, terminated with a slash (/).

1. Name of the local grid refinement - up to 8 characters.
   A local grid name root ending in an asterisk (*) may be used to refer to several local grids in the same record.

2. Number of active wells required to keep the refinement open.
   If set > 0, the refinement is on whenever it contains the stated number of active wells or more, and off whenever the number of active wells falls below the stated number.

   DEFAULT: 0, which switches the refinement off unconditionally.

If the LGR is part of an amalgamation specified using the keyword AMALGAM, then the whole amalgamation is switched off (that is all local grids contained in the amalgamation are switched off). The number of active wells specified by item 2 refers to the amalgamation (that is it applies to all wells within local grids contained in the amalgamation).

In addition, refer to the OPTIONS keyword, item 56.

See also keyword LGRON.

---

**Examples**

**Example 1**

This switches the refinement SOUTH off unconditionally:

```
LGROFF
SOUTH /
```

**Example 2**

The refinement NORTH is on whenever it has 2 or more active wells, and off whenever it has less than 2:

```
LGROFF
-- NAME NWELLS
NORTH 2 /
```
Switches a local grid refinement on

LGRON can be used at any time during the run to switch a local grid refinement on. Optionally, the local grid refinement can be switched on and off automatically depending on the number of active wells it contains. (A well is active if it is open or stopped, but not shut.)

The keyword is followed by either one or two items of data, terminated with a slash (/).

1. Name of the local grid refinement - up to 8 characters.
   
   A local grid name root ending in an asterisk (*) may be used to refer to several local grids on the same record.

2. Number of active wells required to keep the refinement open.
   
   If set > 0, the refinement will be on whenever it contains the stated number of active wells or more, and off whenever the number of active wells falls below the stated number.
   
   DEFAULT: 0, which switches the refinement on unconditionally.

In the timestep immediately following a LGRON keyword, all local grids start off with a one-day timestep.

If the LGR is part of an amalgamation specified using the keyword AMALGAM, then the whole amalgamation is switched on (that is all local grids contained in the amalgamation are switched on). The number of active wells specified by item 2 refers to the amalgamation (that is it applies to all wells within local grids contained in the amalgamation).

See also keyword LGROFF.

Note: Switching a local grid refinement on part way through a run is not advised, in general, since there is no valid procedure for allocating material to refined cells except at initial conditions when equilibration is used. If a refinement is switched on during a simulation, material is distributed to the refined cells within each host cell according to the pore volumes of the refined cells. In particular, for a run employing the vertical equilibrium model, there is no way to set the contact depths correctly while accounting for oil immobilization effects. Neither can the effects of hysteresis be properly accounted for when a local grid is switched on during a simulation.

Note: This keyword has no effect in parallel runs, when an LGR is switched off (using the LGROFF keyword), it cannot be switched on again.

Examples

Example 1

This switches the refinement SOUTH on unconditionally:

```
LGRON 'SOUTH' /
```

Example 2

The refinement NORTH is on whenever it has 2 or more active wells, and off whenever it has less than 2:
LGRON

-- NAME       NWELLS
   'NORTH'   2      /
**Reserve ECLIPSE 100 option licenses**

This keyword may optionally be used to reserve ECLIPSE 100 option FLEX licenses at the beginning of the run, rather than waiting until a keyword activating the option is encountered. For certain options the activating keyword is not encountered until the SCHEDULE section, perhaps part way through a run, and it is possible that the run could be stopped at that point if an appropriate license is not available.

The data consists of one or more records, each containing the name of an ECLIPSE 100 option terminated by a slash (/). The set of records must be terminated by a blank record containing just a slash.

**Note:** This keyword is used by the Simulation launchers to work out which licenses are used (except for "parallel") in order to implement the license aware job scheduling for ECLIPSE 100 and ECLIPSE 300. In such case, the aliases below should not be used.

Please refer to the appropriate documentation in the ECLIPSE distribution, which details which license names are supported.

The option names (with aliases in italic) are:

<table>
<thead>
<tr>
<th>LICENSES option</th>
<th>ECLIPSE option</th>
</tr>
</thead>
<tbody>
<tr>
<td>'eclipse4d'</td>
<td>The Petro-elastic Model</td>
</tr>
<tr>
<td>'friction'</td>
<td>The Wellbore Friction Model</td>
</tr>
<tr>
<td>'gasfield'</td>
<td>Gas Field Operations Model</td>
</tr>
<tr>
<td>'gaslift'</td>
<td>Gas Lift Optimization</td>
</tr>
<tr>
<td>'lgr' or 'lgrs'</td>
<td>Local Grid Refinement and Coarsening</td>
</tr>
<tr>
<td>'multiseg' or 'multisegwells'</td>
<td>Multisegment Wells</td>
</tr>
<tr>
<td>'network' or 'networks'</td>
<td>The Network Option</td>
</tr>
<tr>
<td>'unencodedhmd'</td>
<td>Allow unencoded sensitivities output from the Gradient option.</td>
</tr>
</tbody>
</table>

*Table 3.19: LICENSES options*

Each option name can be in upper or lower case, and must be spelled out fully. Abbreviations are not recognized.

For the 2014.1 release of ECLIPSE, a number of license options were retired. If the LICENSES keyword is used with the following options, a warning message will be issued advising that the option has been retired.

- CO₂ Storage
- Coal Bed Methane Model
- Environmental Tracers
- Flux Boundary Conditions
- Foam Model
- Gas Calorific Value Control
• Polymer Flood Model
• Gi Pseudo-Compositional Model
• Solvent Model
• Surfactant Model

**Example**

```plaintext
LICENSES
'network' /
'lgr' /
'friction' /
```

**Keywords**

1249
LIFTOPT

Turns on gas lift optimization

The LIFTOPT keyword activates the gas lift optimization facility (see “Gas lift optimization” of the ECLIPSE Technical Description), and sets parameters governing the lift gas increment size, the minimum economic gradient, and the frequency of optimization calculations. This keyword must be entered before any of the other keywords associated with the facility, namely GLIFTOPT and WLIFTOPT.

The keyword is followed by a line containing between two and four items, ending with a slash (/).

1. Increment size for lift gas injection rate
   Lift gas is allocated to individual wells in whole numbers of the increment size.
   If gas lift optimization is no longer required, it can be turned off by entering a zero or negative number.
   UNITS: sm\(^3\)/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm\(^3\)/day (PVT–M)

2. Minimum economic gradient of improvement in oil production rate per unit increase in lift gas injection rate
   Further lift gas increments will not be allocated to a well if the resulting increase in oil production rate, multiplied by the well’s weighting factor, divided by the lift gas increment size, is less than the minimum economic gradient.
   UNITS: sm\(^3\)/sm\(^3\) (METRIC), stb/Mscf (FIELD), scc/scc (LAB), sm\(^3\)/sm\(^3\) (PVT–M)

3. Minimum interval between gas lift optimizations
   This item determines the frequency at which subsequent gas lift optimizations are performed. The distribution of lift gas is optimized at each timestep that starts after the specified interval has elapsed since the previous optimization.
   If there is a production network, gas lift can only be optimized when the network is being balanced.
   The minimum interval between network balances is set in item 1 of keyword NETBALAN.
   UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT–M)
   DEFAULT: 0.0 (which will cause an optimization to be performed every timestep)

4. Optimize gas lift during each of the first NUPCOL iterations of the timestep?
   YES The lift gas distribution is optimized during each of the first NUPCOL Newton iterations of the timestep. This enables flow targets and limits to be observed more accurately. (The value of NUPCOL can be set using keyword NUPCOL or GCONTOL). During any remaining iterations, the lift gas allocation will remain constant so as not to hinder their convergence.
   NO Optimization is only performed in the first Newton iteration of the timestep. This takes less time than the YES option, but if reservoir conditions change appreciably over the timestep then rate targets and limits may not be obeyed accurately.

ECLIPSE 100 If there is a production network (see “Network option” in the ECLIPSE Technical Description), optimization can only be performed when the network is being balanced. Thus, the YES option to optimize in each of the first NUPCOL Newton iterations can only be used if the network is also being balanced with this frequency. (Setting a negative balancing interval in item 1 of keyword NETBALAN causes the network to be balanced in each of the first NUPCOL Newton iterations.) If the network is balanced only in the first iteration, the NO option is used regardless of the entry here.
DEFAULT: YES
End the data record with a slash (/).

Example

<table>
<thead>
<tr>
<th>LIFTOPT</th>
<th>minimum</th>
<th>optimization</th>
<th>opt in 1st</th>
</tr>
</thead>
<tbody>
<tr>
<td>-- increment</td>
<td>gradient</td>
<td>interval</td>
<td>NUPCOL its?</td>
</tr>
<tr>
<td>-- size</td>
<td>SM3/DAY</td>
<td>SM3/SM3</td>
<td>DAYS</td>
</tr>
<tr>
<td>4E4</td>
<td>5E-3</td>
<td>0.0</td>
<td>YES</td>
</tr>
</tbody>
</table>
Limits for Li-correlation

This keyword takes two items of data, terminated with a slash (/), for each EoS region:

1. Lower limit for hydrocarbon-water interpolation $f_g$
   
   DEFAULT: 0.75

2. Upper limit for hydrocarbon water interpolation $f_o$
   
   DEFAULT: 1.25

These values are used to calculate a water-hydrocarbon relative permeability for single phase hydrocarbon states. Oil-gas and gas-oil relative permeabilities are only relevant if both oil and gas are present. This water-hydrocarbon $K_{rh}$ must tend to the user entered $K_{ro}$ when the composition is oil-like, and to $K_{rg}$ when it is gas-like.

To set up a variable to interpolate between these two values, we define the pseudo critical temperature $T_{crit}$, given by the Li correlation as:

$$T_{crit} = \frac{\sum_j T_{cj} \cdot V_{cj} \cdot z_j}{\sum_j V_{cj} \cdot z_j}$$  \hspace{1cm} \text{Eq. 3.109}

Then define $f$ as:

$$f = \frac{T_{crit}}{T_{res}}$$

$f$ will clearly be 1 if the reservoir temperature is the critical temperature, but become greater than 1 for an oil ($T_{crit} > T_{res}$) and less than 1 for a gas. If we choose values at which the system is to be regarded completely as an oil or a gas as $f_o$ (1.25 by default) and $f_g$ (0.75 by default), then an interpolating function may be defined as

$$E = \frac{f \cdot f_g}{f_o \cdot f_g} \quad \text{Eq. 3.110}$$

in the region $f_g < f < f_o$, and 0 or 1 outside this range. This goes from 0 when the system is gas-like ($f$ below $f_g$) to 1 when the system is oil-like ($f$ above $f_o$), with a linear interpolation in between.

See "Near critical oil and gas relative permeabilities" in the ECLIPSE Technical Description for further information.

Example

LILIM

0.97 1.03 /

Keywords

1252
LINCOM

Coefficients for linear combination of phases

This keyword sets the coefficients for a linear combination of oil, water and gas. This keyword must be specified if you wish to set a linearly combined production rate target or use a linear combination of the phases to specify a guide rate. For example, the volume of a quantity, \( V_s \), may be equal to

\[
V_s = \alpha V_{oil} + \beta V_{wat} + \gamma V_{gas}
\]  \hspace{1cm} \text{Eq. 3.111}

and the flow rate of this quantity, \( Q_s \), equal to

\[
Q_s = \alpha Q_{oil} + \beta Q_{wat} + \gamma Q_{gas}
\]  \hspace{1cm} \text{Eq. 3.112}

for coefficients \( \alpha, \beta, \gamma \). These coefficients can be specified using this keyword and then production rate targets may be specified for \( Q_s \). The units of \( V_s \) can be specified with keyword \texttt{LCUNIT} (a default value UNIT is assumed). The units of \( Q_s \) are assumed to be those of \( V_s \) per day (\texttt{METRIC, FIELD, PVT-M}) or per hour (\texttt{LAB}).

The keyword is followed by a single line of data terminated with a slash (/).

1. Oil coefficient, \( \alpha \). This quantity can be a user defined argument (UDA) and can be set with keywords \texttt{UDQ} and \texttt{UDADIMS}.

   \textbf{UNITS:} UNIT/sm\(^3\) (\texttt{METRIC}), UNIT/stb (\texttt{FIELD}), UNIT/scf (\texttt{LAB}), UNIT/sm\(^3\) (\texttt{PVT-M}). Default value UNIT can be changed with \texttt{LCUNIT}.

   \textbf{DEFAULT:} 0.0

2. Water coefficient, \( \beta \). This quantity can be a user defined argument (UDA) and can be set with keywords \texttt{UDQ} and \texttt{UDADIMS}.

   \textbf{UNITS:} UNIT/sm\(^3\) (\texttt{METRIC}), UNIT/stb (\texttt{FIELD}), UNIT/scf (\texttt{LAB}), UNIT/sm\(^3\) (\texttt{PVT-M}). Default value UNIT can be changed with \texttt{LCUNIT}.

   \textbf{DEFAULT:} 0.0

3. Gas coefficient, \( \gamma \). This quantity can be a user defined argument (UDA) and can be set with keywords \texttt{UDQ} and \texttt{UDADIMS}.

   \textbf{UNITS:} UNIT/sm\(^3\) (\texttt{METRIC}), UNIT/Mscf (\texttt{FIELD}), UNIT/scf (\texttt{LAB}), UNIT/sm\(^3\) (\texttt{PVT-M}). Default value UNIT can be changed with \texttt{LCUNIT}.

   \textbf{DEFAULT:} 0.0

\textbf{Note:} This keyword must be specified before any linearly combined rate targets or limits are defined.

Example

\begin{verbatim}
LINCOM 0.5 0.5 0.002 /
\end{verbatim}
**LINKPERM**

**Applies grid block permeabilities to cell faces**

The keyword may optionally be used to instruct ECLIPSE to treat the input grid block permeabilities as applying to the cell faces. ECLIPSE will then use the face permeabilities directly as linking permeabilities in the transmissibility calculation, instead of calculating the linking permeabilities as an average of the adjacent grid block permeabilities.

The keyword should be followed by up to three identifiers, to tell ECLIPSE which face (+ or -) to apply the grid block permeability to in each direction (I, J and K). For any directions not specified, the permeabilities will be treated as block center permeabilities in the usual manner.

The identifier ‘I+’ indicates that PERMX or PERMR should be applied to the face between blocks (I, J, K) and (I+1, J, K). Alternatively, the identifier ‘I-’ applies the permeability to the face between blocks (I, J, K) and (I-1, J, K).

The identifiers ‘J+’ and ‘J-’ apply similarly to PERMY or PERMTHT, while ‘K+’ and ‘K-’ imply that permeabilities input by PERMZ should refer to corresponding block faces in the Z direction.

The data must be terminated by a slash (/).

The options specified by LINKPERM for each direction override those specified by PERMAVE.

The use of linking permeabilities is described in “Transmissibility calculations” in the ECLIPSE Technical Description.

**Examples**

**Example 1**

Conventional geometry:

Permeabilities are input by PERMX for I+ block faces, PERMY for block centers and PERMZ for K- block faces.

```
LINKPERM I+ K- /
```

**Example 2**

Radial geometry:

Permeabilities are input using PERMTHT for THETA+ block faces, while PERMR and PERMZ are defaulted to block center values.

```
LINKPERM J+ /
```
The live oil option is to be used

This keyword can only be used with the THERMAL option. It allows hydrocarbon components to partition in the liquid (oil) and gas phases. This option is the default when the THERMAL keyword is specified in the RUNSPEC section.

This keyword has no associated data.
Scaled low salinity (oil-wet) end point oil relative permeabilities

The LKRO keyword specifies the value of oil relative permeability at the maximum oil saturation within each grid block for scaling the low salinity (oil-wet) oil relative permeability saturation tables.

The LKRO keyword should only be used if oil is present in the model and if both the end-point scaling and low salinity options have been selected via the RUNSPEC keywords ENDScale and LOWSALT respectively.

The LKRO keyword should be followed by one real number for each grid block in the current input box specifying the maximum oil relative permeability. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LKRO is omitted for some or all of the grid blocks in a model where it could be specified, the oil relative permeabilities at maximum oil saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling” in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```
-- In the RUNSPEC section :
DIMENS
   1  7  3  /
ENDSCALE
/
-- In the PROPS section :
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
   1   1   2   7   1   3
LKRO
   6*0.55
   6*0.65
   6*0.67 /
```
LKRORG

Scaled low salinity (oil-wet) end point oil relative permeabilities

The LKRORG keyword specifies the value of oil relative permeability at the critical gas saturation within each grid block for scaling the low salinity (oil-wet) oil relative permeability saturation tables.

The LKRORG keyword should only be used if oil and gas are present in the model and if both the end-point scaling and low salinity options have been selected via the RUNSPEC keywords ENDScale and LOWSALT respectively.

The LKRORG keyword should be followed by one real number for each grid block in the current input box specifying the oil relative permeability at the critical gas saturation. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LKRORG is omitted for some or all of the grid blocks in a model where it could be specified, the oil relative permeabilities at critical gas saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling” in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```plaintext
-- In the RUNSPEC section:
DIMENS
  1  7  3   /
ENDSCALE
/

-- In the PROPS section :
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX       1   1   2   7   1   3
LKRORG
  6*0.8
  6*0.8
  6*0.7   /
```

Keywords

1257
Scaled low salinity (oil-wet) end point oil relative permeabilities

The LKRORW keyword specifies the value of oil relative permeability at the critical water saturation within each grid block for scaling the low salinity (oil-wet) oil relative permeability saturation tables.

The LKRORW keyword should only be used if oil and water are present in the model and if both the end-point scaling and low salinity options have been selected via the RUNSPEC keywords ENDSCALE and LOWSALT respectively.

The LKRORW keyword should be followed by one real number for each grid block in the current input box specifying the oil relative permeability at the critical water saturation. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LKRORW is omitted for some or all of the grid blocks in a model where it could be specified, the oil relative permeabilities at critical water saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```plaintext
-- In the RUNSPEC section:
DIMENS
  1  7  3 /
ENDSCALE
/

-- In the PROPS section:
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1  1  2  7  1  3

LKRORW
  6*0.5
  6*0.6
  6*0.6 /
```
Scaled low salinity (oil-wet) end point water relative permeabilities

The **LKRW** keyword specifies the value of water relative permeability at the maximum water saturation within each grid block for scaling the low salinity (oil-wet) water relative permeability saturation tables.

The **LKRW** keyword should only be used if water is present in the model and if both the end-point scaling and low salinity options have been selected via the **RUNSPEC** keywords **ENDSCALE** and **LOWSALT** respectively.

The **LKRW** keyword should be followed by one real number for each grid block in the current input box specifying the maximum water relative permeability. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword **LKRW** is omitted for some or all of the grid blocks in a model where it could be specified, water relative permeabilities at maximum water saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the **ECLIPSE Technical Description** for details of the end-point scaling procedure.

**Example**

```plaintext
-- In the RUNSPEC section:
DIMENS
  1 7 3 /
ENDSCALE
/
-- In the PROPS section:
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3
LKRW
  6*0.55
  6*0.65
  6*0.67 /
```
Scaled low salinity (oil-wet) end point water relative permeabilities

The LKRWR keyword specifies the value of water relative permeability at the critical oil saturation within each grid block for scaling the low salinity (oil-wet) water relative permeability saturation tables.

The LKRWR keyword should only be used if water is present in the model and if both the end-point scaling and low salinity options have been selected via the RUNSPEC keywords ENDScale and LOWSALT respectively.

The LKRWR keyword should be followed by one real number for each grid block in the current input box specifying the water relative permeability at the critical oil saturation. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LKRWR is omitted for some or all of the grid blocks in a model where it could be specified, the water relative permeabilities at critical oil saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```plaintext
-- In the RUNSPEC section:
DIMENS
  1  7  3  /  ENDScale
  /
-- In the PROPS section:
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1  1  2  7  1  3
LKRWR
  6*0.5
  6*0.6
  6*0.6  /
```
**LOAD**

### Load a SAVE file to perform a fast restart

The **LOAD** keyword must be used when performing a fast restart run. It tells ECLIPSE to read the **SAVE** file generated by the original run. The **SAVE** file contains processed information from the **RUNSPEC**, **GRID**, **EDIT**, **PROPS** and **REGIONS** sections of the original data file, and these sections should be omitted from the data file of the restart run. Thus the **LOAD** keyword should be the **first** keyword in the data file.

See also keywords **SAVE** and **RESTART**.

The **LOAD** keyword should be followed by one line of data containing the following items:

1. **The root name of the SAVE file to be loaded**
   (Enclosed in quotes.)

2. **The report step that the run should start from.**

   **Note:** If the run which generated the file referred to in item 1 had the **RPTRST** mnemonic **BASIC=6**, then each timestep in that run will have generated save file information. In this case, the report number that needs to be entered here is the timestep number. This information can be obtained from the print file for that run.

   **ECLIPSE 100**

   In ECLIPSE 100 this method of specifying the step at which the runs is to start from is entirely optional. If it is used (this data item is not defaulted) then ECLIPSE 100 will expect to find the time dependent restart record on the **SAVE** file, see the **SAVE** keyword or the **SAVE** argument in the **RPTRST** keyword. To restart from a standard Restart file the **RESTART** keyword should be used in the **SOLUTION** section and this data item defaulted.

3. **The NOSIM switch for the RESTART run.**

   **SIM**  Perform a normal simulation run

   **NOSIM**  Perform a data checking run only

   **DEFAULT:** **SIM**

4. **The formatted/unformatted status of the SAVE file to be read in**

   (enclosed in quotes)

   **FORMATTED** or **UNFORMATTED**

   **DEFAULT:** **UNFORMATTED**

   **ECLIPSE 100 only**

5. **A switch to request a SAVE file to be output (enclosed in quotes).**

   **YES** or **NO**

   **DEFAULT:** **NO**

   Terminate the record with a slash (/).

The **LOAD** keyword is used in place of the **RUNSPEC**, **GRID**, **EDIT**, **PROPS** and **REGIONS** sections. It passes control to the **SOLUTION** section (there is no need to use the **SOLUTION** keyword), where the **RESTART** keyword should be supplied to tell ECLIPSE to read the **RESTART** file. The remainder of the data file should specify the **SUMMARY** and **SCHEDULE** section data.
Fast restarts are not supported for parallel runs.

In ECLIPSE 300, if the SAVE file is unified or formatted (that is UNISAVE or FMTSAVE used in original run), then UNIFIN or FMTIN should be placed before the LOAD keyword. Note that ECLIPSE 100 does not support multiple SAVE files.

The SAVE file switch (item 5) needs to be set to YES if the subsequent Restart files are to be written to the SAVE file. (See the SAVE argument in the RPTRST keyword.)

The pre-2000A ECLIPSE 100 format of the LOAD keyword (without the report step data item) is still recognized, effectively defaulting the report step.

It is not possible to load a SAVE file from a previous version of ECLIPSE; the SAVE file must be created using the same version of ECLIPSE.

Examples

Example 1

A typical ECLIPSE 100 fast restart data file has this form.

```plaintext
-- Typical ECLIPSE 100 case, restarting from a SAVE
-- file and a RESTART file.
--
LOAD 'BASE' /
RESTART 'BASE' 11 /
SCHEDULE :
: :
TSTEP 30.0 /
END
```

Example 2

A typical ECLIPSE 300 restart from a SAVE file.

```plaintext
-- Typical ECLIPSE 300 case, restarting from a SAVE file
--
LOAD 'CASE1' 12 /
SCHEDULE :
: :
DATES 1 'FEB' 1990 /
/
END
```

Example 3

An example NOSIM case in ECLIPSE 100.

```plaintext
-- Formatted NOSIM case in ECLIPSE 100
```
LOAD 'CASE1' 1* 'NOSIM' 'FORMATTED' /
RESTART 'CASE1' 12 /
RPTSOL 1 0 1 1 /
SUMMARY ALL
SCHEDULE :
:
DATES 1 'FEB' 1990 /

END
LOOTUS

Requests run summary output to be in Lotus format

This keyword requests that the run summary output, generated by using the RUNSUM keyword, should be written in a format that can be easily imported into a spreadsheet program such as Lotus 1-2-3.

If this keyword is used, the run summaries are output to a separate file, rather than appended to the end of the print file. (There is no need to use the keyword SEPARATE with LOTUS.) The separate run summary file name is the root with the extension RSM.

Output is in text format, with spaces to separate columns. The file can be imported into Lotus 1-2-3 by using the File-Import-Structured sequence from within the spreadsheet.

The LOTUS keyword has no associated data.

Example

```
LOTUS
```
## LOWSALT

### Enables modeling of low salinity effects in the Brine model

This keyword enables the modeling of low salt water. It will automatically turn on the `BRINE` option if this keyword is not already present.

See "Brine tracking" in the *ECLIPSE Technical Description* for further information.

The `LOWSALT` keyword has no associated data.
LPCW

Scaled low salinity (oil-wet) maximum oil-water capillary pressure

The LPCW keyword specifies the maximum value of oil-water capillary pressure for scaling the low-salinity (oil-wet) water-oil capillary pressure saturation tables.

The LPCW keyword should only be used if water and oil are present in the model and if both the end-point scaling and low salinity options have been selected via the RUNSPEC keywords ENDScale and LOWSALT respectively.

The LPCW keyword should be followed by one real number for each grid block in the current input box specifying the maximum oil-water capillary pressure. The data must be terminated by a slash (/).

UNITs: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LPCW is omitted for some or all of the grid blocks in a model where it could be specified, the maximum oil-water capillary pressures at connate water saturation will default to the values defined in the appropriate capillary pressure tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Notes

• The LPCW keyword should not be used in combination with the J-function option specified by the keywords JFUNC or JFUNCR because the J-function calculation is used to define the water capillary pressure scaling and any values defined by the input LPCW keyword will be ignored. In this case the maximum capillary pressure for each cell is reported in the INIT file.

• The LPCW values used to scale the oil-water capillary pressure must be consistent with the capillary pressure values in the tables. In particular both must possess the same sign.

Example

```
-- In the RUNSPEC section:
DIMENS   1  7  3  /
ENDSCALE /
-- In the PROPS section:
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX      1  1  2  7  1  3
LPCW     6*10.0
          6*12.5
          6*15.0 /
```
**LSALTFNC**

**Tables of low-salt weighting factors versus salt concentration**

This keyword is set to input the weighting factors for the low-salinity saturation functions as a function of the salt concentration. The data comprises NTSFUN (item 1 of keyword TABDIMS) tables, each terminated by a slash (/). These coefficients are used in calculating the saturation end points, the water and oil relative permeabilities and the water-oil capillary pressure when the LOWSALT option is active.

Each table consists of 3 columns of data:

1. **Salt concentration**
   - Note that the effective salinity from the multi-component brine model (keyword ECLMC) will not be used to provide salinity values for this table; ECLMC is incompatible with this keyword.
   - The values should increase monotonically down the column.
   - UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/cc (LAB), kg/sm$^3$ (PVT-M)

2. **Weighting factor** $F_1$ for the low-salinity saturation endpoints and the relative permeabilities interpolation.
   - The values should decrease monotonically down the column.
   - A value of 0.0 implies that only the high-salinity saturation functions will be used (tables defined by SATNUM) and a value of 1.0 implies that only low-salinity saturation functions will be used (tables defined by LWSLTNUM).
   - The value should not be greater than 1.0 or less than 0.0.

3. **Weighting factor** $F_2$ for the low-salinity capillary pressure interpolation.
   - The values should decrease monotonically down the column.
   - A value of 0.0 implies that only the high-salinity saturation functions will be used (as defined by SATNUM) and a value of 1.0 implies that only low-salinity saturation functions will be used (as defined by LWSLTNUM).
   - The value should not be greater than 1.0 or less than 0.0.
   - This item may be defaulted and is then taken as the value in the second column.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (item 3 of keyword TABDIMS).

**Note:** NTSFUN tables must be supplied; however the LWSLTNUM table number is used. A defaulted table after the first one is copied from the previous table.

**Note:** When the SURFACTW option is active, the weighting factors in a LWSLTNUM table are used to interpolate between the immiscible oil-wet high-salinity curves (SATNUM tables) and immiscible oil-wet low-salinity curves (LWSLTNUM tables). The same factors are used to interpolate between the immiscible water-wet high-salinity curves (tables defined by SURFWNUM) and the immiscible water-wet low-salinity curves (tables defined by LSLTWNUM).
Example

With NT SFUN =4 and NSSFUN >2 specified in keyword TABDIMS:

```
   LSALTFNC
   --unused table
   0   1.0 1*
   20  0.0 1*
   -- Second table copied from previous table
   -- factors used for salinity interpolation in region 1
   /--unused table
   -- factors used for salinity interpolation in region 2
   0   1.0 1*
   30  0.0 1*
   /--REGIONS----------------------------------------------
   SATNUM
   150*1 150*3 /
   LWSLTNUM
   150*2 150*4 /
```
LSCRIT

Linear solution convergence criteria

This is an optional keyword which modifies the convergence criteria used to control the linear iterations of the WARP linear solver. The WARP linear solver is initiated using the WARP keyword. See "Nested factorization for LGRs" in the ECLIPSE Technical Description for more details.

Specific criteria may be assigned to individual local grids or sub-grids using keyword LSCRITS, which will override the criteria set here.

The keyword should be followed by some or all of the following items, terminated by a slash (/).

1. MXOR Maximum number of orthogonal vectors used in the GMRES linear solver for all local grids.
   DEFAULT: 0
2. NIT Maximum number of linear iterations for all local grids.
   DEFAULT: 0
3. TOL Required improvement in linear solver RMS residual for all local grids.
   DEFAULT: 1.0E-3

Example

```
LSCRIT
  5 5 1.0E-2 /
```

Keywords

ECLIPSE 100
ECLIPSE 300
SPECIAL
RUNSPEC
GRID
EDIT
PROPS
REGIONS
SOLUTION
SUMMARY
SCHEDULE

ECLIPSE Reference Manual

1269
LSCRITS  Linear solution convergence criteria for individual sub-grids

This is an optional keyword which modifies the convergence criteria used to control the linear iterations on individual sub-grids within the WARP linear solver (activated with keyword WARP). The area covered by a sub-grid can best be seen by visualizing the output from the SUBG mnemonic in the RPTRST keyword.

The keyword should be followed one or more records. Each record is terminated by a slash (/). The set of records must be terminated by an empty record with a single slash.

1. 'NAME' If the name of a local grid is entered, the convergence criteria in the following items are applied to all sub-grids within this local-grid.

   If the string 'BASE' is entered, the convergence criteria in the following items are applied to all sub-grids within the base grid.

   If the string does not match an LGR name (or 'BASE'), the first 3 letters of the string are used to identify the group of sub-grids to which the convergence criteria are to be applied:

   '---C' The rest of the 8-character string is interpreted as a color index. The convergence criteria in the following items are applied to all sub-grids with this color. The color of sub-grids is best visualized using the output from the COLR mnemonic in the RPTRST keyword.

   '---S' The rest of the 8-character string is interpreted as a sub-grid index. The convergence criteria in the following items are applied to the sub-grid with this index. The sub-grids within WARP are best visualized using the output from the SUBG mnemonic in the RPTRST keyword.

2. MXOR Maximum number of the orthogonal vectors used in the GMRES linear solver for this local grid.
   DEFAULT: 0

3. NIT Maximum number of linear iterations for this sub-grid.
   DEFAULT: 0

4. TOL Required improvement in linear solver RMS residual for this sub-grid.
   DEFAULT: 1.0E-3

See "The ECLIPSE 300 WARP linear solver" in the ECLIPSE Technical Description.

Note: The index to individual sub-grids will change as a function of the number of processors set in the PARALLEL keyword, or the coloring strategy specified in the WCOL keyword.

Example

```
LSCRITS
  BASE 10 10 1.0E-4 / --set base grid
  --C3 1 1 1.0E-3 / --set color 3
  --L2 3 3 1.0E-4 / --set level 2 refinement
  rad01 40 40 1.0E-7 / --set lgr rad01
  rad02 40 40 1.0E-7 / --set lgr rad02
  rad03 40 40 1.0E-7 / --set lgr rad03
/```

Keywords

1270
**LSLTWNUM**  **Low-salt water-wet saturation function region numbers**

The keyword should be followed by one integer for every grid block specifying the low-salinity water-wet saturation function region to which it belongs. The region number should not be less than 1 or greater than NTSFUN (see item 1 of keyword **TABDIMS**). The data must be terminated by a slash (/).

The low salinity water-wet saturation function region number specifies the saturation table (input using **SWFN**, **SOF3** and related keywords, in the **PROPS** section) to be used to calculate relative permeabilities and capillary pressures in each grid block.

The water-wet curves are computed as a weighted average between the low salinity water-wet saturation functions (**LSLTWNUM** region number) and the high salinity water-wet saturation functions (**SURFWNUM** region number). The low-salinity weighting factors are taken from the **LWSLTNUM** table in the **LSALTFNC** keyword.

This keyword can be used only if the **SURFACTW** and **LOWSALT** keywords have been entered in the **RUNSPEC** section to initialize the modeling of the low salinity effects in the brine model, and the surfactant change to wettability.

Grid blocks are ordered with the X-axis index cycling fastest, followed by the Y- and Z-axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

---

**Note:** The **LSLTWNUM** keyword or its alias **LWSNUM** must be specified if the **LOWSALT** and **SURFACTW** options have been specified.

---

**Example**

With NTSFUN=4 specified in keyword **TABDIMS** and NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in **RUNSPEC** keyword **DIMENS**:

```
  LSLTNUM
  144*3 96*4 /
```

---

**Keywords**

1271
LSNUM

Low salinity (oil-wet) saturation function region numbers

The keyword should be followed by one integer for every grid block specifying the low-salinity saturation function region to which it belongs. The region number should not be less than 1 or greater than NTSFUN (see item 1 of keyword TABDIMS). The data must be terminated by a slash (/).

The low salinity oil-wet saturation function region number specifies the saturation table (input using SWFN, SOF3 and related keywords in the PROPS section) to be used to calculate relative permeabilities and capillary pressures in each grid block.

The oil-wet curves are computed as a weighted average between the low salinity oil-wet saturation functions (LSNUM region number) and the high salinity oil-wet saturation functions (SATNUM region number). The low-salinity weighting factors are taken from the LSNUM region specified tables in the LSALTFNC keyword.

This keyword can be used only if the LOWSALT keyword has been entered in the RUNSPEC section.

When the Surfactant option is active, the LSNUM region numbers correspond to the immiscible low salinity curves. For the SURFACTW option, this corresponds to the immiscible oil-wet low salinity curves. The SATNUM region numbers correspond to the immiscible high salinity curves or to the immiscible oil-wet high salinity curves if the SURFACTW option is active.

Grid blocks are ordered with the X-axis index cycling fastest, followed by the Y- and Z-axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

Note: The LSNUM keyword is an alias for the LWSLNUM keyword. One of these keywords must be specified if the LOWSALT option has been specified.

Example

With NTSFUN=2 specified in keyword TABDIMS and NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS:

| LSNUM | 144*1 96*2 / |

Keywords
LSOGCR

Scaled low salinity (oil-wet) critical oil-in-gas saturations

The LSOGCR keyword specifies the critical oil in gas saturation (that is, the largest oil saturation for which the oil relative permeability to gas is zero) within each grid block for scaling the low salinity (oil-wet) oil relative permeability to gas saturation tables.

The tabulated oil relative permeability to gas functions are scaled between the critical oil in gas saturation defined by the LSOGCR keyword and the maximum oil saturation defined in terms of the minimum water and gas saturations using the LSWL and SGL keywords respectively.

The LSOGCR keyword should only be used if gas is present in the model and if both the end-point scaling and low salinity options have been selected via the RUNSPEC keywords ENDScale and LOWSALT respectively.

The LSOGCR keyword should be followed by one real number for each grid block in the current input box specifying the critical oil in gas saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LSOGCR is omitted for some or all of the grid blocks in a model where it could be specified, the critical oil in gas saturations for the low salinity oil relative permeability to gas calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```
-----RUNSPEC section
DIMENS
1 7 3 /
ENDSCALE
/
-----PROPS section
--------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
1 1 2 7 1 3
LSOGCR
5*0.24 10*0.21 3*0.20 /
```
LSOWCR  

**Scaled low salinity (oil-wet) critical oil-in-water saturations**

The LSOWCR keyword specifies the critical oil in water saturation (that is, the largest oil saturation for which the oil relative permeability to water is zero) within each grid block for scaling the low salinity (oil-wet) oil relative permeability to water saturation tables.

The tabulated oil relative permeability to water functions are scaled between the critical oil in water saturation defined by the LSOWCR keyword and the maximum oil saturation defined in terms of the minimum water and gas saturations using the LSWL and SGL keywords respectively.

The LSOWCR keyword should only be used if water is present in the model and if both the end-point scaling and low salinity options have been selected via the RUNSPEC keywords ENDScale and LOWSALT respectively.

The LSOWCR keyword should be followed by one real number for each grid block in the current input box specifying the critical oil in water saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LSOWCR is omitted for some or all of the grid blocks in a model where it could be specified, the critical oil in water saturations for the low salinity oil relative permeability to water calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

### Example

```
------RUNSPEC section
DIMENS
1 7 3 /
ENDSCALE
/
------PROPS section
--------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
 1 1 2 7 1 3
LSOWCR
5*0.21 10*0.19 3*0.18 /
```
LSWCR Scaled low salinity (oil-wet) critical water saturations

The LSWCR keyword specifies the critical water saturation (that is, the largest water saturation for which the water relative permeability is zero) within each grid block for scaling the low salinity (oil-wet) water relative permeability saturation tables.

The tabulated water relative permeability functions are scaled between the critical water saturation defined by the LSWCR keyword and the maximum water saturation defined by the LSWU keyword.

The LSWCR keyword should only be used if water is present in the model and if both the end-point scaling and low salinity options have been selected via the RUNSPEC keywords ENDScale and LOWSALT respectively.

The LSWCR keyword should be followed by one real number for each grid block in the current input box specifying the critical water saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LSWCR is omitted for some or all of the grid blocks in a model where it could be specified, the critical water saturations for the low salinity water relative permeability calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling” in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```
-----RUNSPEC section
DIMENS
  1 7 3 /
ENDSCALE
/
-----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3 /
LSWCR
  5*0.22 10*0.17 3*0.19 /
ENDBOX
```
Scaled low salinity (oil-wet) connate water saturations

The LSWL keyword specifies the connate water saturation (that is, the smallest water saturation in a water saturation function table) within each grid block for scaling the low salinity (oil-wet) oil-water capillary pressure and oil relative permeability to water tables.

The tabulated oil-water capillary pressure functions are scaled linearly between the connate water saturation defined using the LSWL keyword and the maximum water saturation defined using the LSWU keyword.

The LSWL keyword is also used for scaling the oil relative permeability curves.

If the connate water saturation has additionally been specified using the LSWLPF keyword, this saturation end-point will be used instead for scaling the oil-water capillary pressure curves. The connate water saturation entered using LSWL will then only be used to scale the oil relative permeability curves.

The LSWL keyword should be followed by one real number for each grid block in the current input box specifying the connate water saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LSWL is omitted for some or all of the grid blocks in a model where it could be specified, the connate water saturations for low salinity (oil-wet) oil relative permeability and capillary pressure calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```plaintext
-----RUNSPEC section
DIMENS
  1 7 3 /
ENDSCALE
/
-----PROPS section
---------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3
LSWL
  5*0.22 10*0.17 3*0.19 /
```
LSWLPC  Scaled low salinity (oil-wet) connate water saturations, for Pc curves only

The LSWLPC keyword specifies the connate water saturation (that is, the smallest water saturation in a water saturation function table) within each grid block for scaling the low salinity (oil-wet) oil-water capillary pressure tables.

The tabulated oil-water capillary pressure functions are scaled linearly between the connate water saturation defined using the LSWLPC keyword and the maximum water saturation defined using the LSWU keyword.

The LSWLPC keyword is not used for scaling the oil relative permeability curves.

The LSWLPC keyword should only be used if water is present in the model and if both the end-point scaling and low salinity options have been selected via the RUNSPEC keywords ENDSCALE and LOWSALT respectively.

The LSWLPC keyword should be followed by one real number for each grid block in the current input box specifying the connate water saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LSWLPC is omitted for some or all of the grid blocks in a model where it could be specified, the connate water saturations for the low salinity water-oil capillary pressure calculations will default to the values defined by the LSWL keyword.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```plaintext
----RUNSPEC section
DIMENS
1 7 3 /
ENDSCALE
/
----PROPS section
LSWLPC
7*0.22 7*0.20 7*0.18 /
```
LSWU

Scaled low salinity (oil-wet) saturation table
maximum water saturations

The LSWU keyword specifies the maximum water saturation (that is, the highest water saturation in a water saturation function table) within each grid block for scaling the low salinity (oil-wet) oil-water capillary pressure and water relative permeability saturation tables.

The maximum water saturation is used to scale both the oil-water capillary pressure and water relative permeability curves. The tabulated capillary pressure curves are linearly scaled between the connate water saturation defined using the LSWL or LSWLPC keywords and the maximum water saturation defined using the LSWU keyword. The tabulated water relative permeability curves are scaled between the critical water saturation defined using the LSWCR keyword and the maximum water saturation defined using the LSWU keyword.

The LSWU keyword should only be used if water is present in the model and if both the end-point scaling and low salinity options have been selected via the RUNSPEC keywords ENDSATEL and LOWSALT respectively.

The LSWU keyword should be followed by one real number for each grid block in the current input box specifying the maximum water saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LSWU is omitted for some or all of the grid blocks in a model where it could be specified, the maximum water saturations for the low salinity water relative permeability and capillary pressure calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

-----RUNSPEC section
DIMENS
 1 7 3 /
ENDSCALE
/
-----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
 1 1 2 7 1 3
LSWU
18*1.00 /
Calculate Sigma from Lx, Ly, Lz

This keyword is optional in dual porosity runs with the viscous displacement option active (keyword VISC in the RUNSPEC section). It enables ECLIPSE to calculate the sigma factor from the input matrix block dimensions Lx, Ly and Lz.

The keyword should be followed by some or all of the following items, to allow SIGMA to be calculated from Lx, Ly, Lz. Item 4 is used to calculate SIGMAGD from Lz if the gravity drainage option is active.

The data must be terminated by a slash (/).

Items 1 to 3: \( f_x, f_y, f_z \) factors in the Sigma calculation. DEFAULT: 4.0

item 4: \( f_{gd} \) factor in the Sigmagd calculation. DEFAULT: 0.0

item 5: Specify the transmissibility option to use

XONLY Use the X-permeability only (see below)

ALL Use all three directional permeabilities

DEFAULT: XONLY

This sigma factor is calculated from the matrix block size using the expression:

\[
\Sigma = \frac{f_x}{l_x^2} + \frac{f_y}{l_y^2} + \frac{f_z}{l_z^2}
\]

Eq. 3.113

where

\( l_x, l_y \) and \( l_z \) are x, y and z dimensions of the matrix blocks entered using the LX, LY and LZ keywords.

(Any zero (defaulted) values of \( l_x, l_y \) or \( l_z \) do not contribute to this sum.)

The Sigmagd factor is calculated from the matrix block height using the expression:

\[
\Sigma_{gd} = \frac{f_{gd}}{l_z^2}
\]

Eq. 3.114

or zero when \( l_z = 0.0 \) (defaulted).

If item 5 is defaulted (XONLY), the Sigma factor is used directly in the matrix-to-fracture transmissibility calculations. In this case the transmissibility is given by

\[
TR = CDARCY \cdot K \cdot V \cdot \sigma
\]

Eq. 3.115

where K is taken as the X-permeability in the matrix.

Setting item 5 to ALL modifies the calculation of the transmissibility to utilize the matrix permeability in all three directions:

\[
TR = CDARCY \cdot V \cdot \left( \frac{f_x \cdot K_x}{l_x^2} + \frac{f_y \cdot K_y}{l_y^2} + \frac{f_z \cdot K_z}{l_z^2} \right)
\]

Eq. 3.116
**Note:** Any values of SIGMA (V) and SIGMAGD (V) entered in the GRID section are ignored if the keyword is used.

See also the LX, LY, LZ, SIGMA and SIGMAGD keywords.

**Example**

```
LTOSIGMA
  4.0 4.0 0.0 /
```
LTRACE  Force a streamline trace for the Lagrangian tracer solver option

The LTRACE keyword contains no items. When this keyword is encountered, streamlines will be retraced at the end of the next timestep.

**Example**

```
LTRACE
```
LUMPDIMS  Activates and dimensions the LUMPING option

The data consists of two items, describing the numbers of lumped and detailed components (see "Delumping and lumping" in the ECLIPSE Technical Description). The data must be terminated by a slash (/).

1. The number of lumped components
2. The maximum number of detailed components per lumped component.

Example

Five lumped components; each having six detailed components

```
LUMPDIMS
5   6  /
```
LUMPING

Set up names of tracers describing detailed composition of a lumped hydrocarbon component

This keyword sets up a number of tracers for a lumped simulation (keyword LUMPDIMS in the RUNSPEC section), associating each of them with a particular lumped component used in the study.

The keyword should be followed by one line for each lumped hydrocarbon component in the simulation. Each line must contain the data items listed below, and is terminated by a slash (/). The set of records must end with a blank record, containing only a slash (/)

Record 1  ‘Lumped’ hydrocarbon component name
                Each name must correspond to those given in the CNAMES keyword.

Record 2  The number of detailed components in this lumped component, NDET

Record 3  Name given to the tracer representing the first detailed component of item 1

...  

3+NDET-1  Name given to the tracer representing the NDET\textsuperscript{th} detailed component of item 1

\textbf{CAUTION:} The order in which these tracer names are entered is very important, as it is implicitly used by the DETAILVD keyword when calculating the mole fractions of the initial distribution of the detailed components, and by other keywords defining detailed component properties.

\textbf{Note:} It is not necessary to have a detailed representation of every component in the ‘lumped’ simulation.

Example

Two lumped components, having two and three detailed components respectively:

```plaintext
LUMPING
C1 2 z1 z2/
C2 3 z3 z4 z5/
/
```

Keywords

ECLIPSE Reference Manual

1283
**LWKRO**

**Scaled low salinity water-wet end point oil relative permeabilities**

The LWKRO keyword specifies the value of oil relative permeability at the maximum oil saturation within each grid block for scaling the low salinity water-wet oil relative permeability saturation tables.

The LWKRO keyword should only be used if oil is present in the model and if the end-point scaling, low salinity and surfactant wettability options have been selected via the RUNSPEC keywords ENDScale, LOWSALT and SURFACTW respectively.

The LWKRO keyword should be followed by one real number for each grid block in the current input box specifying the maximum oil relative permeability. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LWKRO is omitted for some or all of the grid blocks in a model where it could be specified, the oil relative permeabilities at maximum oil saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

**Example**

```plaintext
-- In the RUNSPEC section:
DIMENS
  1  7  3  /
ENDSCALE
/

-- In the PROPS section:
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1  1  2  7  1  3

LWKRO
  6*0.55
  6*0.65
  6*0.67  /
```

Keywords
**LWKORG**

**Scaled low salinity water-wet end point oil relative permeabilities**

The **LWKORG** keyword specifies the value of oil relative permeability at the critical gas saturation within each grid block for scaling the low salinity water-wet oil relative permeability saturation tables.

The **LWKORG** keyword should only be used if oil and gas are present in the model and if the end-point scaling, low salinity and surfactant wettability options have been selected via the **RUNSPEC** keywords **ENDSCALE**, **LOWSALT** and **SURFACTW** respectively.

The **LWKORG** keyword should be followed by one real number for each grid block in the current input box specifying the oil relative permeability at the critical gas saturation. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword **LWKORG** is omitted for some or all of the grid blocks in a model where it could be specified, the oil relative permeabilities at critical gas saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the *ECLIPSE Technical Description* for details of the end-point scaling procedure.

**Example**

```
-- In the RUNSPEC section :
DIMENS
  1  7  3  /
ENDSCALE
/

-- In the PROPS section :
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1   1   2   7   1   3
LWKORG
6*0.8
6*0.8
6*0.7 /
```
LWKJORW  Scaled low salinity water-wet end point oil relative permeabilities

The LWKJORW keyword specifies the value of oil relative permeability at the critical water saturation within each grid block for scaling the low salinity water-wet oil relative permeability saturation tables.

The LWKJORW keyword should only be used if oil and water are present in the model and if the end-point scaling, low salinity and surfactant wettability options have been selected via the RUNSPEC keywords ENDScale, LOWSALT and SURFACTW respectively.

The LWKJORW keyword should be followed by one real number for each grid block in the current input box specifying the oil relative permeability at the critical water saturation. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LWKJORW is omitted for some or all of the grid blocks in a model where it could be specified, the oil relative permeabilities at critical water saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```
-- In the RUNSPEC section :
DIMENS
  1 7 3 /
ENDSCALE
/
-- In the PROPS section :
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
   1 1 2 7 1 3
LWKJORW
  6*0.5
  6*0.6 /
```
The **LWKRW** keyword specifies the value of water relative permeability at the maximum water saturation within each grid block for scaling the low salinity water-wet water relative permeability saturation tables.

The **LWKRW** keyword should only be used if water is present in the model and if the end-point scaling, low salinity and surfactant wettability options have been selected via the **RUNSPEC** keywords **ENDSCALE**, **LOWSALT** and **SURFACTW** respectively.

The **LWKRW** keyword should be followed by one real number for each grid block in the current input box specifying the maximum water relative permeability. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword **LWKRW** is omitted for some or all of the grid blocks in a model where it could be specified, water relative permeabilities at maximum water saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the **ECLIPSE Technical Description** for details of the end-point scaling procedure.

### Example

```plaintext
-- In the RUNSPEC section :
DIMENS
  1  7  3  /
ENDSCALE
/
-- In the PROPS section :
--------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3
LWKRW
  6*0.55
  6*0.65
  6*0.67 /
```

### Keywords

```
x  ECLIPSE 100
  ECLIPSE 300
  SPECIAL
  RUNSPEC
  GRID
  EDIT
  x  PROPS
  REGIONS
  SOLUTION
  SUMMARY
  SCHEDULE
```
LWKRWR  Scaled low salinity water-wet end point water relative permeabilities

The LWKRWR keyword specifies the value of water relative permeability at the critical oil saturation within each grid block for scaling the low salinity water-wet water relative permeability saturation tables.

The LWKRWR keyword should only be used if water is present in the model and if the end-point scaling, low salinity and surfactant wettability options have been selected via the RUNSPEC keywords ENDScale, LOWSalt and SURFACTW respectively.

Each LWKRWR keyword should be followed by one real number for each grid block in the current input box specifying the water relative permeability at the critical oil saturation. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LWKRWR is omitted for some or all of the grid blocks in a model where it could be specified, the water relative permeabilities at critical oil saturation will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```
-- In the RUNSPEC section :
DIMENS
  1 7 3 /
ENDSCALE /
-- In the PROPS section :
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX :
    1 1 2 7 1 3
LWKRWR
  6*0.5
  6*0.6
  6*0.6 /
```
LWPCW

Scaled low salinity water-wet maximum oil-water capillary pressure

The LWPCW keyword specifies the maximum value of oil-water capillary pressure for scaling the low salinity water-wet water-oil capillary pressure saturation tables.

The LWPCW keyword should only be used if water and oil are present in the model and if the end-point scaling, low salinity and surfactant wettability options have been selected via the RUNSPEC keywords ENDScale, LOWSALT and SURFACTW respectively.

The LWPCW keyword should be followed by one real number for each grid block in the current input box specifying the maximum oil-water capillary pressure. The data must be terminated by a slash (/).

UNITs: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LWPCW is omitted for some or all of the grid blocks in a model where it could be specified, the maximum oil-water capillary pressures at connate water saturation will default to the values defined in the appropriate capillary pressure tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Notes

- The LWPCW keyword should not be used in combination with the J-function option specified by the keywords JFUNC or JFUNCR because the J-function calculation is used to define the water capillary pressure scaling and any values defined by the input LPCW keyword will be ignored. In this case the maximum capillary pressure for each cell is reported in the INIT file.

- The LWPCW values used to scale the oil-water capillary pressure must be consistent with the capillary pressure values in the tables. In particular both must possess the same sign.

Example

```
-- In the RUNSPEC section :
DIMENS
  1  7  3 /
ENDSCALE /
-- In the PROPS section :
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX       1  1  2  7  1  3
LWPCW
  6*10.0
  6*12.5
  6*15.0 /
```

Keywords

1289
LWSLTNUM  Low-salt oil-wet saturation function region numbers

The keyword should be followed by one integer for every grid block specifying the low-salinity saturation function region to which it belongs. The region number should not be less than 1 or greater than NTSFUN (see item 1 of keyword TABDIMS). The data must be terminated by a slash (/).

The low salinity oil-wet saturation function region number specifies the saturation table (input using SWFN, SOF3 and related keywords in the PROPS section) to be used to calculate relative permeabilities and capillary pressures in each grid block.

The oil-wet curves are computed as a weighted average between the low salinity oil-wet saturation functions (LWSLTNUM region number) and the high salinity oil-wet saturation functions (SATNUM region number). The low-salinity weighting factors are taken from the LWSLTNUM table in the LSALTFNC keyword.

This keyword can be used only if the LOWSALT keyword has been entered in the RUNSPEC section to initialize the modeling of the low salinity effects in the Brine Model.

When the surfactant option is active, the LWSLTNUM region numbers correspond to the immiscible low salinity curves. For the SURFACTW option, it corresponds to the immiscible oil-wet low salinity curves. The SATNUM region numbers correspond to the immiscible high salinity curves or to the immiscible oil-wet high salinity curves if the SURFACTW option is active.

Grid blocks are ordered with the X-axis index cycling fastest, followed by the Y- and Z-axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

Note: The LWSLTNUM keyword or its alias LSNUM must be specified if the LOWSALT option has been specified.

Example

With NTSFUN=2 specified in keyword TABDIMS and NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS:

```
LWSLTNUM
144*1 96*2 /
```
**LWSNUM  Low salinity water-wet saturation function region numbers**

The keyword should be followed by one integer for every grid block specifying the low-salinity water-wet saturation function region to which it belongs. The region number should not be less than 1 or greater than NTSFUN (see item 1 of keyword **TABDIMS**). The data must be terminated by a slash (/).

The low salinity water-wet saturation function region number specifies the saturation table (input using **SWFN**, **SOF3** and related keywords in the **PROPS** section) to be used to calculate relative permeabilities and capillary pressures in each grid block.

The water-wet curves are computed as a weighted average between the low salinity water-wet saturation functions (LWSNUM region number) and the high salinity water-wet saturation functions (HWSNUM region number). The low-salinity weighting factors are taken from the LNUM region specified tables in the **LSALTFNC** keyword.

This keyword can be used only if the **LOWSALT** and **SURFACTW** keywords have been entered in the **RUNSPEC** section and corresponds to the immiscible low salinity water-wet curves. The HWSNUM region numbers correspond to the immiscible high salinity water-wet curves.

Grid blocks are ordered with the X-axis index cycling fastest, followed by the Y- and Z-axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

---

*Note:* The LWSNUM keyword is an alias for the LSLTWNUM keyword. One of these keywords must be specified if the **LOWSALT** and **SURFACTW** options have been specified.

---

**Example**

With NTSFUN=2 specified in keyword **TABDIMS** and NDIVX = 8, NDIVY=6 and NDIVZ=5 specified in **RUNSPEC** keyword **DIMENS**:

```
LWSNUM
144*1 96*2 /
```
**LWSOGCR**  
**Scaled low salinity water-wet critical oil-in-gas saturations**

The LWSOGCR keyword specifies the critical oil in gas saturation (that is, the largest oil saturation for which the oil relative permeability to gas is zero) within each grid block for scaling the low salinity water-wet oil relative permeability to gas saturation tables.

The tabulated oil relative permeability to gas functions are scaled between the critical oil in gas saturation defined by the LWSOGCR keyword and the maximum oil saturation defined in terms of the minimum water and gas saturations using the LSWL and SGL keywords respectively.

The LWSOGCR keyword should only be used if gas is present in the model and if the end-point scaling, low salinity and surfactant wettability options have been selected via the RUNSPEC keywords ENDScale, LOWSALT and SURFACTW respectively.

The LWSOGCR keyword should be followed by one real number for each grid block in the current input box specifying the critical oil in gas saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LWSOGCR is omitted for some or all of the grid blocks in a model where it could be specified, the critical oil in gas saturations for the low salinity water-wet oil relative permeability to gas calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

**Example**

```
----RUNSPEC section
DIMENS
 1 7 3 /
ENDSCALE
/
----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3
LWSOGCR
  5*0.24 10*0.21 3*0.20 /
```
Scaled low salinity water-wet critical oil-in-water saturations

The **LWSOWCR** keyword specifies the critical oil in water saturation (that is, the largest oil saturation for which the oil relative permeability to water is zero) within each grid block for scaling the low salinity water-wet oil relative permeability to water saturation tables.

The tabulated oil relative permeability to water functions are scaled between the critical oil in water saturation defined by the **LWSOWCR** keyword and the maximum oil saturation defined in terms of the minimum water and gas saturations using the **LSWL** and **SGL** keywords respectively.

The **LWSOWCR** keyword should only be used if water is present in the model and if the end-point scaling, low salinity and surfactant wettability options have been selected via the **RUNSPEC** keywords **ENDSCALE**, **LOWSALT** and **SURFACTW** respectively.

The **LWSOWCR** keyword should be followed by one real number for each grid block in the current input box specifying the critical oil in water saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword **LWSOWCR** is omitted for some or all of the grid blocks in a model where it could be specified, the critical oil in water saturations for the low salinity water-wet oil relative permeability to water calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the **ECLIPSE Technical Description** for details of the end-point scaling procedure.

**Example**

```plaintext
-----RUNSPEC section
DIMENS
  1 7 3 /
ENDSCALE
/
-----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3
LWSOWCR
  5*0.21 10*0.19 3*0.18 /
```
Scaled low salinity water-wet critical water saturations

The **LWSWCR** keyword specifies the critical water saturation (that is, the largest water saturation for which the water relative permeability is zero) within each grid block for scaling the low salinity water-wet water relative permeability saturation tables.

The tabulated water relative permeability functions are scaled between the critical water saturation defined by the **LWSWCR** keyword and the maximum water saturation defined by the **LWSWU** keyword.

The **LWSWCR** keyword should only be used if water is present in the model and if the end-point scaling, low salinity and surfactant wettability options have been selected via the **RUNSPEC** keywords **ENDSCALE**, **LOWSALT** and **SURFACTW** respectively.

The **LWSWCR** keyword should be followed by one real number for each grid block in the current input box specifying the critical water saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword **LWSWCR** is omitted for some or all of the grid blocks in a model where it could be specified, the critical water saturations for the low salinity water-wet water relative permeability calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the **ECLIPSE Technical Description** for details of the end-point scaling procedure.

**Example**

```
----RUNSPEC section
DIMENS
  1 7 3 /
ENDSCALE
/
----PROPS section
------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3 /
LWSWCR
  5*0.22 10*0.17 3*0.19 /
ENDBOX
```
Scaled low salinity water-wet connate water saturations

The **LWSWL** keyword specifies the connate water saturation (that is, the smallest water saturation in a water saturation function table) within each grid block for scaling the low salinity water-wet oil-water capillary pressure and oil relative permeability to water tables.

The tabulated oil-water capillary pressure functions are scaled linearly between the connate water saturation defined using the **LWSWL** keyword and the maximum water saturation defined using the **LWSWU** keyword.

The **LWSWL** keyword is also used for scaling the oil relative permeability curves.

If the connate water saturation has additionally been specified using the **LWSWLPC** keyword, this saturation end-point will be used instead for scaling the oil-water capillary pressure curves. The connate water saturation entered using **LWSWL** will then only be used to scale the oil relative permeability curves.

The **LWSWL** keyword should only be used if water is present in the model and if the end-point scaling, low salinity and surfactant wettability options have been selected via the **RUNSPEC** keywords **ENDSCALE**, **LOWSALT** and **SURFACTW** respectively.

The **LWSWL** keyword should be followed by one real number for each grid block in the current input box specifying the connate water saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword **LWSWL** is omitted for some or all of the grid blocks in a model where it could be specified, the connate water saturations for low salinity water-wet oil relative permeability and capillary pressure calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the *ECLIPSE Technical Description* for details of the end-point scaling procedure.

**Example**

```
-----RUNSPEC section
DIMENS
  1 7 3 /
ENDSCALE
/
-----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3
LWSWL
  5*0.22 10*0.17 3*0.19 /
```
LWSWLPC  Scaled low salinity water-wet connate water saturations, for Pc curves only

The LWSWLPC keyword specifies the connate water saturation (that is, the smallest water saturation in a water saturation function table) within each grid block for scaling the low salinity water-wet oil-water capillary pressure tables.

The tabulated oil-water capillary pressure functions are scaled linearly between the connate water saturation defined using the LWSWLPC keyword and the maximum water saturation defined using the LWSWU keyword.

The LWSWLPC keyword is not used for scaling the oil relative permeability curves.

The LWSWLPC keyword should only be used if water is present in the model and if the end-point scaling, low salinity and surfactant wettability options have been selected via the RUNSPEC keywords ENDScale, LOWSALT and SURFACTW respectively.

The LWSWLPC keyword should be followed by one real number for each grid block in the current input box specifying the connate water saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LWSWLPC is omitted for some or all of the grid blocks in a model where it could be specified, the connate water saturations for the low salinity water-wet capillary pressure calculations will default to the values defined by the LWSWL keyword.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```plaintext
-----RUNSPEC section
DIMENS
  1 7 3 /
ENDSCALE
/
-----PROPS section
LWSWLPC
  7*0.22   7*0.20   7*0.18 /
```
LWSWU

Scaled low salinity water-wet saturation table
maximum water saturations

The LWSWU keyword specifies the maximum water saturation (that is, the highest water saturation in a water saturation function table) within each grid block for scaling the low salinity water-wet oil-water capillary pressure and water relative permeability saturation tables.

The maximum water saturation is used to scale both the oil-water capillary pressure and water relative permeability curves. The tabulated capillary pressure curves are linearly scaled between the connate water saturation defined using the LWSWL or LWSWLPC keywords and the maximum water saturation defined using the LWSWU keyword. The tabulated water relative permeability curves are scaled between the critical water saturation defined using the LWSWCR keyword and the maximum water saturation defined using the LWSWU keyword.

The LWSWU keyword should only be used if water is present in the model and if the end-point scaling, low salinity and surfactant wettability options have been selected via the RUNSPEC keywords ENDScale, LOWSALT and SURFACTW respectively.

The LWSWU keyword should be followed by one real number for each grid block in the current input box specifying the maximum water saturation. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword LWSWU is omitted for some or all of the grid blocks in a model where it could be specified, the maximum water saturations for the low salinity water-wet water relative permeability and capillary pressure calculations will default to the values defined in the appropriate relative permeability tables.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```plaintext
-----RUNSPEC section
DIMENS
   1 7 3 /
ENDSCALE
/
-----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX           1 1 2 7 1 3
LWSWU
   18*1.00 /
```

Keywords
**X direction matrix block sizes for the viscous displacement option**

The keyword is optional in dual porosity runs with the viscous displacement option active (keyword `VISCD` in the `RUNSPEC` section). The keyword defines the representative matrix block size in the X direction.

The keyword should be followed by one positive real number for every grid block in the current input box. The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example `115*208.4`). Note that spaces must not be inserted on either side of the asterisk.

If this keyword is not present when the viscous displacement option is active, the matrix block size in the X direction is taken as zero in the calculation of \( \sigma_{V} \) when the `LTOSIGMA` keyword is present.

**UNITS:** m (METRIC), ft (FIELD), cm (LAB)

See also keywords `LY`, `LZ` and `LTOSIGMA`.

### Example

```
BOX
  3 16 3 8 1 1 /
LX
  98*10.0 /
```
Y direction matrix block sizes for the viscous displacement option

The keyword is optional in dual porosity runs with the viscous displacement option active (keyword VISCD in the RUNSPEC section). The keyword defines the representative matrix block size in the Y direction.

The keyword should be followed by one positive real number for every grid block in the current input box. The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

If this keyword is not present when the viscous displacement option is active, the matrix block size in the Y direction is taken as zero in the calculation of the viscous displacement term and does not contribute towards the calculation of SIGMAV when the LTOSIGMA keyword is present.

UNITS: m (METRIC), ft (FIELD), cm (LAB)

See also keywords LX, LZ and LTOSIGMA.

Example

<table>
<thead>
<tr>
<th>BOX</th>
<th>1 100 1 100 1 3 /</th>
</tr>
</thead>
<tbody>
<tr>
<td>LY</td>
<td>10000*10.0</td>
</tr>
<tr>
<td></td>
<td>10000*20.0</td>
</tr>
<tr>
<td></td>
<td>10000*30.0 /</td>
</tr>
</tbody>
</table>

Keywords

1299
Z direction matrix block sizes for the viscous displacement option

The keyword is optional in dual porosity runs with the viscous displacement option active (keyword VISCD in the RUNSPEC section). The keyword defines the representative matrix block size in the Z direction.

The keyword should be followed by one positive real number for every grid block in the current input box. The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

If this keyword is not present when the viscous displacement option is active, the matrix block size in the Z direction is taken as zero in the calculation of the viscous displacement term and does not contribute towards the calculation of SIGMAV or SIGMAGDV when the LTOSIGMA keyword is present.

UNITS: m (METRIC), ft (FIELD), cm (LAB)

See also keywords LX, LY and LTOSIGMA.

Example

```
BOX 1 5 5 1 1 /
LZ 5*40.0 5*1.5 15*40.0 /
```
This section contains an alphabetic listing of keywords starting with the letter M.
MAPAXES  

Input of pre-processor map origin

This keyword is entirely optional.

The keyword actually specifies grid axes and the grid origin relative to the map coordinates. It is normally output by grid pre-processors so that the origin of the maps used to generate the grid can be stored. The origin is then available through the Grid file for post-processing.

In figure 3.9, the map coordinates are a right-handed system with the z-axis pointing up. The grid is laid out on the map; its axes form a right-handed system with the z-axis pointing down, using the ECLIPSE convention, and therefore appear as a left-handed system in the diagram.

The grid axes are specified by three points: a point on the grid Y-axis, the grid origin, and a point on the X-axis. They are given in this order to conform to the right-handed map coordinate system used.

Following the keyword line, there is a single record containing six items of data, ending with a slash (/):

1. X1 The X coordinate of one point of the grid Y-axis relative to the map
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
2. Y1 The Y coordinate of one point of the grid Y-axis relative to the map
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
3. X2 The X coordinate of the grid origin relative to the map
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
4. Y2 The Y coordinate of the grid origin relative to the map
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
5. X3 The X coordinate of one point of the grid X-axis relative to the map
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
6. Y3 The Y coordinate of one point of the grid X-axis relative to the map
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
See also the \texttt{MAPUNITS} keyword.

\textbf{Example}

\begin{verbatim}
MAPAXES 2700 3900 3000 4000 3100 3700 /
\end{verbatim}
**MAPUNITS**

**Specifies units used for MAPAXES data**

This keyword is entirely optional. It is normally output by grid pre-processors.

The MAPUNITS keyword defines the units used for the MAPAXES data, which defines the simulation grid position with respect to maps.

Following the keyword line, there is a single record containing one item of data, ending with a slash (/).

**CAUTION:** Use the European spelling METRES: the US spelling METERS is not recognized.

**Example**

```
MAPUNITS
METRES /
```

The units are then available through the GRID file for post-processing.

See also the MAPAXES keyword.
This keyword is used to define the upstream massflow boundary condition of a river as a function of the time.

The keyword is followed by `NMMAST` tables with a maximum of `MXMAST` entries in each. These numbers are set with the `RIVRdims` keyword. Each table is terminated with a slash (/).

Each table consists of 5 columns of data:
1. Day
2. The name of the month abbreviated to three characters
   - JAN, FEB, MAR, APR, MAY, JUN, JUL, AUG, SEP, OCT, NOV or DEC.
   - JUL is an acceptable alternative to JLY.
3. Year input should be a 4 digit integer.
4. Time of observation in 24 hour clock format, hh:mm
   - Do not use 24:00; use 00:00 of the following day.
5. Massflow at this time
   - UNITS: m$^3$/day (METRIC), ft$^3$/day (FIELD), cc/hr (LAB)

There is a minimum of two rows of data per table. End the table with a slash (/). After the first tables subsequent tables may be defaulted to the previous table by just entering a slash.

The times, items 1-3 considered as a whole, must be monotonically increasing down the table.

**Example**

```
MASSFLOW
1  JAN 1995  00:00  1589.0
  8  FEB 1995 2:45  2010.0
15  SEP 1995 12:00  307.0
/
```
**MATCORR**

**Initiates the material correction option**

The material correction option can be used to reduce the accumulated material balance error in a simulation. In some cases the facility may allow the simulation to run more quickly. The application of the technique when the non-linear equations are not fully converged should only be tried with great care.

The application of material correction is not available in conjunction with the parallel option.

The keyword is followed by the following items of data, all of which are normally defaulted. The data must be terminated with a slash (/).

1. Newton iteration number after which the correction can be applied, provided the solution has reached the convergence criteria supplied in items 2 and 3.
   
   DEFAULT: 12

2. Non-linear convergence error.
   
   DEFAULT: 0.01

   
   DEFAULT: 1E-6

Following a timestep, the material balance correction subtracts any material balance error from the residual of the subsequent timestep. Hence the material balance error at one timestep is carried over and resolved at the next timestep. This method leads to very small accumulated material balance errors.

The correction is applied either when the timestep has converged, or earlier when the input data items allow.

**Examples**

**Example 1**

Apply the material balance correction.

```
MATCORR /
```

**Example 2**

Apply the material balance correction after Newton iteration 2, provided the solution has converged to a looser than normal tolerance.

```
MATCORR 3 0.1 1E-6 /
```
MAXTRANZ Maximum z-transmissibility

The keyword should be followed by a single positive real number, the maximum allowed z-transmissibility between two cells. Where calculated values are larger than this value, they are reduced.

UNITS: cP.rm$^3$/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm$^3$/day/atm (PVT-M)

DEFAULT: 1.0E20 (all unit sets)

Example

```
MAXTRANZ
20000 /
```
**MAXVALUE**  
Apply a maximum limit to an array in the current box

The keyword may be followed by any number of records, each of which is terminated by a slash (/). The data is terminated by a null record (that is a record with no data before the terminating slash). Each record consists of at least two items of data.

1. The name of the array to be modified.  
2. The maximum value for the array specified by item 1.  
   
   The constant should not be negative, but may be real or integer.

   The following data items may be used to redefine the input box for this and subsequent operations within the current keyword. If these items are not defined (a slash is inserted after item 2), they default to the values that were used for the previous operation within the current keyword. For the first operation in the keyword, the box defaults to the values set by the most recent **BOX** or **ENDBOX** keyword. If there is no preceding **BOX** or **ENDBOX** in the current section, the box is taken to include the entire reservoir.

3. **IX1** First block to be modified on the X axis.  
4. **IX2** Last block to be modified on the X axis.  
5. **JY1** First block to be modified on the Y axis.  
6. **JY2** Last block to be modified on the Y axis.  
7. **KZ1** First block to be modified on the Z axis.  
8. **KZ2** Last block to be modified on the Z axis.

The data must satisfy

\[
1 \leq \text{IX1} \leq \text{IX2} \leq \text{NDIVIX} \\
1 \leq \text{JY1} \leq \text{JY2} \leq \text{NDIVIY} \\
1 \leq \text{KZ} \leq \text{KZ2} \leq \text{NDIVIZ} \quad (\text{where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS})
\]

---

**Note:** If the **MAXVALUE** keyword is used before an array has been input, it has no effect on that array when it is subsequently input. **MAXVALUE** can only operate on arrays that are already input.

---

See also keywords **BOX**, **ENDBOX**, **EQUALS**, **ADD**, **MULTIPLY**, **MINVALUE** and **COPY**.

---

**Example**

In the **PROPS** section:

```
--- ARRAY MAX ------ BOX ------
MAXVALUE  'SWL'  0.35   3 8   1 10 3 3 /
'SWCR'  0.40 / defaults to last specified box /
```

---

**Keywords**
MEMORY

Allocates the required memory at start of run

The data consists of 2 integers, the second of which allows the user to change the default memory allocations for character strings that ECLIPSE 100 uses when running. For parallel runs the value is applied to each processor.

ECLIPSE 100 and 300 use dynamic memory. Therefore, this keyword is of little value and should be avoided.

This keyword is ignored by ECLIPSE 300 runs.

The first item is ignored in ECLIPSE 100. The second item allows the imposition of a fixed amount of memory for character strings in ECLIPSE 100. The simulation will stop with an error if this amount is exceeded. By default, ECLIPSE 100 determines a large enough size dynamically.

1. Unused.
2. A fixed amount of character memory in thousands of CHARACTER*8 strings.
   DEFAULT: in ECLIPSE 100 the character storage is allocated to an adequate value at run time.

Example

This run allocates a fixed storage for 20000 (twenty thousand) CHAR*8 strings.

```
MEMORY 1* 2.0 /
```
Indicates use of Up-Front Memory option

The up-front memory option can only be used if you already know the active cell information when building a dataset.

There are 2 different ways this option may be used:

1. The `ACTNUM` keyword and its data must be supplied at the very top of the `GRID` section before the grid properties have been specified.

2. Use the `GDFILE` keyword to import an extensible grid file. Only `.EGRID` files can be imported as this grid file contains the necessary active cell information required to use this option. The `GDFILE` keyword, together with the imported filename, must be specified at the very top of the `GRID` section.

Local grid refinement option. If a deck contains local grid refinements, it is important that the LGR data, including any `ACTNUM` information, be specified very early on in the `GRID` section, otherwise grid property arrays will not be set up properly.

To compute grid properties, such as pore volume and transmissibility, ECLIPSE will focus on the active cells only. This is especially important for large models that have difficulty processing the `GRID` section, and reaching the `PROPS` section.

MEMSAVE contains no items.

Note: Not all runs will benefit from using this keyword. For instance, simulation runs that could benefit are datasets with a low ratio of active cells to total number of grid cells; decks using `EQUALS / ADD / MULTIPLY / COPY` to define grid properties like `PORO`, `PERMX`, for example.

Current restrictions

If any of the following keywords are present in a dataset, then the simulator will turn the up-front memory option off: `AQUDIMS`, `CONDFLTS`, `COAL`, `COARSEN`, `DUALPORO`, `EXTFIN`, `FLUXNUM`, `GEOMECH`, `HMDIMS`, `MPFA`, `NINEPOIN`, `PARALLEL`, `PEBI`, `SOLVDIMS`, `THERMAL`, `USEFLUX`, `VDFLOW`, `VDKRO`, `VDKRG`

For any of the above options, a warning message will be output, and the run will continue as normal.
Specify a message

This enables a message to be defined to the error handling system. This system produces output normally seen on the screen when running interactively, and is echoed to the print and debug files. Examples of messages produced by the program are:

```
@--Message
@ Processing GRID data
@--Message
@ Neither NTG nor DZNET entered
@ All net to gross ratios taken as unity
@--Message
@ Problem has 36 active cells
```

The MESSAGE keyword allows you to define such a one-line message. This can be useful for establishing landmarks in large Print files.

Like the TITLE keyword, the line immediately after the MESSAGE keyword is taken as the message, and no quotes or slash characters are required.

This should not be confused with the MESSAGES keyword, used to set limits on the error handling system.

Example

This defines a message to be displayed.

```
MESSAGE
 Starting CO2 injection phase of run
```

In ECLIPSE 100 this example produces an output message of the form:

```
@--MESSAGE AT TIME 0.0 DAYS (1-JAN-2009):
 @  Starting CO2 injection phase of run
```

In ECLIPSE 300 the output message format is:

```
@--Message
@ Starting CO2 injection phase of run
```
MESSAGES  Resets message print and stop limits

This keyword can be used to reset the print and stop limits for messages of any severity type.

There are 6 levels of severity:

1. Message (Not an error, purely informative)
2. Comment (Probably not a data error)
3. Warning (Possibly a data error)
4. Problem (Calculation difficulties)
5. Error (Definitely a data error)
6. Bug (Suspected programming error).

Printing of a particular type of message ceases after its print limit has been reached. The run stops if a particular type of message is generated more times than its stop limit. However, the run is switched into data checking mode if any error message is generated, so that no timestep calculations are performed after the next \texttt{TIME}, \texttt{TSTEP} or \texttt{DATES} keyword.

The keyword should be followed by some or all of the following integers, terminated by a slash (/). Repeat counts (for example 3*1000) and defaults (for example 2*) can be used if required. Any items defaulted or left unspecified will not be altered. The items are initialized with their default values.

1. Print limit for severity 1 messages
   DEFAULT: 1000000
2. Print limit for severity 2 messages
   DEFAULT: 1000000
3. Print limit for severity 3 messages
   DEFAULT: 10000
4. Print limit for severity 4 messages
   DEFAULT: 100
5. Print limit for severity 5 messages
   DEFAULT: 100
6. Print limit for severity 6 messages
   DEFAULT: 100
7. Stop limit for severity 1 messages
   DEFAULT: 1000000
8. Stop limit for severity 2 messages
   DEFAULT: 1000000
9. Stop limit for severity 3 messages
   DEFAULT: 10000
10. Stop limit for severity 4 messages

Keywords

1312
11. Stop limit for severity 5 messages
   DEFAULT: 10

12. Stop limit for severity 6 messages
   DEFAULT: 1

**ECLIPSE 300**

This item only applies to severity 6 messages issued in the SCHEDULE section.

*Note:* It is not advisable to alter the stop limits for messages of severity 5 and 6.

**ECLIPSE 100**

13. Print limit for groups of frequently output messages. Each group of messages (e.g. PVT table extrapolation warnings, messages about connections between non-neighbor connections between local grid refinements) has a tally (per processor) which is incremented each time a message from the group is output
   DEFAULT: 10

*Note:* Messages duplicated across multiple processors in parallel are only output once in the main PRT file but will still contribute to the message tally for each processor.

**Example**

```
MESSAGES
2* 10 5* 10000 / alters print and stop limits for warnings
```
MESSOPTS Controls severity type for messages reporting timesteps forced to be accepted

This keyword can be used to control the severity type for messages reporting timesteps that are forced to be accepted.

The keyword should be followed by the following items, terminated by a slash (/).

1. Mnemonic ACCPTIME.
2. An integer selecting the severity type:
   1. Report forced timesteps as message.
   2. Report forced timesteps as comment.

In ECLIPSE 100, timesteps that are forced to be accepted are reported by default with problem severity type.

In ECLIPSE 300, timesteps that are forced to be accepted are reported by default with message severity type.

Note: The total number of timesteps forced to be accepted is printed in the PRT file in the summary at the end of a run independently of the use of this keyword.

Note: The use of this keyword may increase the number of occurrences of certain severity type messages. If the number of occurrences of a particular type of messages increases up to reaching the print or stop limit for messages of such severity type, then printing that type of message will cease or the run will stop. The MESSAGES keyword can be used to reset print and stop limits for messages of any severity type.

Example

MESSOPTS ACCPTIME 3 / reports forced timesteps as Warning messages
**METRIC**

**Metric units are to be used**

This indicates that metric units are to be used. Note that this is the default unit convention.

See "Units" in the *ECLIPSE Technical Description* for further details.

The keyword has no associated data.

<table>
<thead>
<tr>
<th>x</th>
<th>ECLIPSE 100</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>ECLIPSE 300</td>
</tr>
<tr>
<td></td>
<td>SPECIAL</td>
</tr>
<tr>
<td></td>
<td>RUNSPEC</td>
</tr>
<tr>
<td></td>
<td>GRID</td>
</tr>
<tr>
<td></td>
<td>EDIT</td>
</tr>
<tr>
<td></td>
<td>PROPS</td>
</tr>
<tr>
<td></td>
<td>REGIONS</td>
</tr>
<tr>
<td></td>
<td>SOLUTION</td>
</tr>
<tr>
<td></td>
<td>SUMMARY</td>
</tr>
<tr>
<td></td>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>
Specify cell mid-point depths

The keyword should be followed by one real number for each grid block in the current input box.

This keyword specifies the depth value of the middle of each cell. This is an alternative to TOPS, the TOPS value being obtained as:

\[ \text{TOPS}_i = \text{MIDS}_i - \frac{DZ_i}{2} \]  

Eq. 3.117

This keyword should not be confused with DEPTH, used in the EDIT section to overwrite the calculated cell center depth values.

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

Example

For a reservoir 20 by 4 by 4 with depth of 5025 ft:

```
MIDS
320*5025 /
```
**MINDZNET**  

**Minimum net thickness for active cell**

The keyword should be followed by a single non-negative real number, the minimum net thickness ($DZ$) allowed for an active cell. If the value is zero, the check will not be performed and no cells will be set inactive because their thickness is too small.

In **THERMAL** runs, cells with net thickness below the minimum, will be treated as rock-filled cells and may still be active.

The data must be followed by a slash.

Cells with a smaller net $DZ$ value are regarded as inactive.

**UNITS:** m (**METRIC**), ft (**FIELD**), cm (**LAB**), m (**PVT-M**)

**DEFAULT:** 0.000001 (all unit sets)

**Example**

```
MINDZNET  
0.1 / 
```
MINNNCT

Sets a cutoff transmissibility below which a non-neighbor connection is deleted

The keyword is optional and is used to declare threshold transmissibility values below which a non-neighbor connection is deleted.

The keyword should be followed by up to three positive real numbers: the cutoff transmissibility, the cutoff diffusivity, and the cutoff thermal transmissibility. Any or all of these can be defaulted.

**UNITS:**

- **Transmissibility**: cP.rm³/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm³/day/atm (PVT-M)
- **Diffusivity**: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
- **Thermal transmissibility**: kJ/day/K (METRIC), Btu/day/°R (FIELD), J/hr/K (LAB), kJ/day/K (PVT-M)

**DEFAULT:** no cutoff

The data must be terminated by a slash (/).

*Note:* If non-neighbor connections are input using the `NNC` keyword, the cutoff is only applied if the `MINNNCT` keyword is specified before the `NNC` keyword.

If the `MINNNCT` keyword is specified with one or more of its arguments defaulted, those values default to zero (no cutoff). However, if the keyword is not specified, the default values are $10^{-6}$ for transmissibility and $10^{-20}$ for diffusivity and thermal transmissibility.

If a non-neighbor connection has one or more of transmissibility, diffusivity and/or thermal transmissibility above the threshold value, but also one or more of them is below the threshold value, the values which are below the threshold value will be reset to zero.

If a model does not use the diffusivity and/or thermal transmissibility options, any values specified for them are ignored.

For some models, setting very small values for the cutoffs, or no cutoff, can result in an extremely large number of non-neighbor connections being generated. This has implications both for speed and memory requirements and is best avoided unless non-neighbor connections with very small transmissibility are significant for the model in question.

**Example**

```
MINNNCT
0.01 2*1E-20 /
```
MINPORV  Set a minimum pore volume a cell must have to be active

This keyword is used to declare a threshold pore volume that a cell must exceed or it will be made inactive. The keyword should be followed by a single positive real number, the minimum pore volume of an active cell, in the current units.

UNITS: \( \text{rm}^3 \) (METRIC), \( \text{rb} \) (FIELD), \( \text{rcc} \) (LAB), \( \text{rm}^3 \) (PVT-M)

DEFAULT: 0.000001 (all unit sets)

The data must be terminated by a slash (/).

The keyword causes any cell with a pore volume less than the input value to become inactive.

An inactive cell does not contribute to the total volume of the system, and is treated, by default, as a barrier. In order for the non-neighbor connections to be established between adjoining cells across an inactive cell, pinchouts must be enabled using the PINCH keyword, and non-neighbor connections must be enabled (which is done by default unless keyword NONNC is encountered in the RUNSPEC section).

In the absence of MINPORV, the default minimum pore volume is 1.0E-6 (in the current units).

See also the related keyword MINPV, and MINPVV in ECLIPSE 100.

MINPV is an alias for MINPORV.

---

**CAUTION:** Cells modified with this keyword will remain inactive unless the ACTNUM value for these cells is set to 1.

---

**Note:** In a THERMAL simulation a cell will only be set inactive if the rock volume is also less than the minimum rock volume specified with the MINROCKV keyword.

---

**Example**

```
MINPORV 0.001 /
```
MINPORV9  Set a minimum pore volume for calculation of nine-point transmissibilities

This keyword is used to declare a threshold pore volume that a cell must exceed to employ nine-point transmissibilities when the MPFA keyword is used to select the nine-point MPFA scheme.

See "Tensor permeability" in the ECLIPSE Technical Description for further information.

The keyword should be followed by a single positive real number, the minimum pore volume in the current units in order to calculate nine-point transmissibilities for this cell.

UNITs: \( \text{rm}^{3} \) (METRIC), \( \text{rb} \) (FIELD), \( \text{rcc} \) (LAB), \( \text{rm}^{3} \) (PVT-M)

DEFAULT: 0.000001 (all unit sets)

The data must be terminated by a slash (/).

The keyword causes any cell with a pore volume less than the input value to use the normal five-point transmissibilities.

In the absence of MINPORV9, the default minimum pore volume is 1.0E-6 (in the current units).

**CAUTION:** Cells modified with this keyword will remain inactive unless the ACTNUM value for these cells is set to 1.

**Example**

```
MINPORV9
0.001 /
```
**MINPV**

**Sets a minimum pore volume a cell must have to be active**

The keyword is used to declare a threshold pore volume that a cell must exceed or it will made inactive.

The keyword should be followed by a single positive real number, the minimum pore volume of an active cell, in the current units.

**UNITS:** $\text{rm}^3$ (*METRIC*), $\text{rb}$ (*FIELD*), $\text{rcc}$ (*LAB*), $\text{rm}^3$ (*PVT-M*)

**DEFAULT** 0.000001 (all unit sets)

The data must be terminated by a slash (/).

The keyword causes any cell with a pore volume less than the input value to become inactive.

An inactive cell does not contribute to the total volume of the system, and is treated by default as a barrier. In order for non-neighbor connections to be established between adjoining cells across an inactive cell, pinchouts must be enabled using the keyword *PINCH* or *PINCHOUT*, and non-neighbor connections must be enabled (which is done by default unless keyword *NONNC* is encountered in the RUNSPEC section).

Cells that have been set inactive using the *ACTNUM* keyword remain so even if their pore volume exceeds the threshold; using *MINPV* only affects active cells.

In the absence of *MINPV*, the default minimum pore volume is 1.0E-6 (in the current units).

In ECLIPSE 100, the sum of the total pore volume removed from the grid is output, along with the total number of cells removed. A list of all the cells that are made inactive can be obtained by setting switch 35 of the *RPTGRID* keyword $> 0$ (mnemonic *MINPV*).

See also keyword *MINPVV*.

**CAUTION:** Cells modified with this keyword will remain inactive unless the *ACTNUM* value for these cells is set to 1.

**Note:** In a THERMAL simulation a cell will only be set inactive if the rock volume is also less than the minimum rock volume specified with the *MINROCKV* keyword.

**Example**

```
MINPV
4500 /
```
Sets minimum pore volumes cells must have to be active

This keyword is used to declare a threshold pore volume that a cell must exceed or it will made inactive.

The keyword should be followed by one positive real number for every cell in the current input box, specifying its threshold pore volume in the current units, terminated with a slash (/).

**UNITS:** \( \text{rm}^3 \) (METRIC), \( \text{rb} \) (FIELD), \( \text{rcc} \) (LAB), \( \text{rm}^3 \) (PVT-M)

**DEFAULT:** 0.000001 (all unit sets)

The keyword causes any cell with a pore volume less than the input value associated with that cell to become inactive.

An inactive cell does not contribute to the total pore volume of the system, and is treated by default as a barrier. In order for non-neighbor connections to be established between adjoining cells across an inactive cell, pinchouts must be enabled using the keyword \text{PINCH} or \text{PINCHOUT}, and non-neighbor connections must be enabled (they are enabled by default unless keyword \text{NONNC} is encountered in the RUNSPEC section).

Cells which have been set inactive via the \text{ACTNUM} keyword remain so even if their pore volume exceeds the threshold; using \text{MINPVV} only affects active cells.

Any minimum pore volumes that are not specified by the end of the GRID section default to 1.0E-6 (in the current units). In the absence of \text{MINPV} or \text{MINPVV}, this is the default for all cells.

A list of all the cells that are made inactive can be obtained by the argument \text{MINPVV} in the RPTGRID keyword.

In ECLIPSE 100, a sum of the total pore volume removed from the grid is output, along with the total number of cells removed (using the MINPVV mnemonic).

See also keyword \text{MINPV}.

**CAUTION:** Cells modified with this keyword will remain inactive unless the ACTNUM value for these cells is set to 1.

**Note:** In a THERMAL simulation a cell will only be set inactive if the rock volume is also less than the minimum rock volume specified with the MINROCKV keyword.

### Example

<table>
<thead>
<tr>
<th>BOX</th>
<th>MINPVV</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1000.0</td>
</tr>
<tr>
<td>2</td>
<td>1000.0</td>
</tr>
<tr>
<td>2</td>
<td>1000.0</td>
</tr>
<tr>
<td>3</td>
<td>1000.0</td>
</tr>
<tr>
<td>2</td>
<td>1000.0</td>
</tr>
<tr>
<td>2</td>
<td>1000.0</td>
</tr>
<tr>
<td>3</td>
<td>1000.0</td>
</tr>
<tr>
<td>2</td>
<td>1000.0</td>
</tr>
<tr>
<td>2</td>
<td>500.0</td>
</tr>
<tr>
<td>3</td>
<td>500.0</td>
</tr>
<tr>
<td>2</td>
<td>500.0</td>
</tr>
<tr>
<td>2</td>
<td>500.0</td>
</tr>
<tr>
<td>3</td>
<td>500.0</td>
</tr>
<tr>
<td>2</td>
<td>500.0</td>
</tr>
<tr>
<td>2</td>
<td>500.0</td>
</tr>
</tbody>
</table>
| 3   | 500.0  | /
MINROCKV  Minimum rock volume for active cell

The keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section).

MINROCKV should be followed by a single positive real number, the minimum rock volume of an active cell. Cells with a smaller rock volume than this are regarded as having zero rock volume. Cells with zero rock and pore volume are regarded as inactive, in thermal simulations.

UNITS: rm$^3$ (METRIC), rb (FIELD), rcc (LAB), rm$^3$ (PVT-M)

DEFAULT: 0.000001 (all unit sets)

See "Thermal option" in the ECLIPSE Technical Description for further information on this option.

**CAUTION:** Cells modified with this keyword will remain inactive unless the ACTNUM value for these cells is set to 1.

**Example**

```plaintext
MINROCKV
0.001 /
```
Minimum rock volume for active cell

This keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section).

MINRV is an alias for MINROCKV.
MINVALUE  Apply a minimum limit to an array in the current box

The keyword may be followed by any number of records, each of which is terminated by a slash (/). Each record consists of at least two items of data. The data is terminated by a null record, a record with no data before the terminating slash (/).

1. The name of the array to be modified.
2. The minimum value for the array specified by item 1.
   The constant should not be negative, but may be real or integer.
   The following items may be used to redefine the input box for this and subsequent operations within the current keyword. If these items are not defined (a slash is inserted after item 2), they default to the values that were used for the previous operation within the current keyword. For the first operation in the keyword, the box defaults to the values set by the most recent BOX or ENDBOX keyword. If there is no preceding BOX or ENDBOX in the current section, the box is taken to include the entire reservoir.
3. IX1 First block to be modified on the X axis.
4. IX2 Last block to be modified on the X axis.
5. JY1 First block to be modified on the Y axis.
6. JY2 Last block to be modified on the Y axis.
7. KZ1 First block to be modified on the Z axis.
8. KZ2 Last block to be modified on the Z axis.

The data must satisfy

\[ 1 \leq IX1 \leq IX2 \leq NDIVIX \]
\[ 1 \leq JY1 \leq JY2 \leq NDIVIY \]
\[ 1 \leq KZ1 \leq KZ2 \leq NDIVIZ \text{ (where } NDIVIX, NDIVIY \text{ and } NDIVIZ \text{ are specified in RUNSPEC keyword DIMENS)} \]

Note: If the MINVALUE keyword is used before an array has been input, it will have no effect on that array when it is subsequently input. MINVALUE can only operate on arrays that are already input.

See also keywords BOX, ENDBOX, EQUALS, ADD, MULTIPLY, MAXVALUE and COPY.

Example

In the PROPS section:

```
--- ARRAY    MAX ----- BOX -----
MINVALUE    'SGU'  0.65    / defaults to whole current input box
            'SGU'  0.8    4*  2  2    / layer 2
/```

Keywords

1325
Miscibility function tables

The **MISC** keyword is required in runs which use the Solvent option in miscible mode (keywords **MISCIBLE** and **SOLVENT** in the RUNSPEC section). See "Miscible flood modeling" and "Solvent model" in the *ECLIPSE Technical Description* for further details of these options.

The data consists of **NTMISC** (see keyword **MISCIBLE**) tables of miscibility functions, which control the transition from miscible to immiscible relative permeability models. Each table must be terminated by a slash.

Each table consists of 2 columns of data:

1. The local solvent fraction ($S_{\text{solvent}} / (S_{\text{solvent}} + S_{\text{gas}})$)
   - The values should increase monotonically down the column and cover the range 0 to 1.
2. The corresponding miscibility on a scale from 0 to 1.
   - Zero corresponds to immiscible relative permeabilities and one to miscible relative permeabilities.
   - The values must span the range [0,1] and should be level or increasing down the column. This means that the first value must be 0 and the last value must be 1.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than **NSMISC** (see keyword **MISCIBLE**).

**Example**

With **NTMISC=1** and **NSMISC≥4**

```plaintext
MISC
0.0  0.0
0.1  0.5
0.3  1.0
1.0  1.0  /
```
When the `MISCIBLE` option is used, this keyword allows the exponent of the surface tension ratio to be changed. This is described in more detail in "Surface tension effects" in the *ECLIPSE Technical Description*.

The keyword contains a single item of data.

**DEFAULT: 0.25**

**Example**

```
MISCEXP
0.34 /
```
Miscible treatment of hydrocarbons

The keyword requests a miscible treatment of hydrocarbon relative permeabilities and capillary pressures. The treatment is different between ECLIPSE 100 and ECLIPSE 300.

In ECLIPSE 100, the MISCIBLE keyword enables the miscible flood model based on the Todd-Longstaff treatment. See "Miscible flood modeling" in the ECLIPSE Technical Description for further information.

In ECLIPSE 300, the MISCIBLE keyword activates dependence of relative permeability and capillary pressure on surface tensions according to the PARACHOR values. See "Surface tension effects" in the ECLIPSE Technical Description for further information. The MISCIBLE keyword should only be used for Compositional runs using ECLIPSE 300, not Black Oil runs.

Enable the miscible gas flood option (ECLIPSE 100)

The keyword should be followed by some or all of the following items, describing the dimensions of the miscible residual oil saturation tables and the method for calculating the upstream relative permeabilities. The data must be terminated by a slash (/).

1. NTMISC
   The number of tables of miscible residual oil saturation versus water saturation, and the maximum number of mixing parameter regions. (See the keywords TLMIXPAR and SORWMIS in the PROPS section).
   DEFAULT: 1

2. NSMISC
   The maximum number of saturation nodes in any table of miscible residual oil saturation versus water saturation (see the keyword SORWMIS in the PROPS section).
   DEFAULT: 20

3. Option for upstreaming hydrocarbon relative permeabilities.
   TWOPOINT  The two-point upstreaming algorithm is to be used to calculate the hydrocarbon component relative permeabilities.
   NONE  The standard single point upstreaming method is used involving only the immediate upstream cell

   DEFAULT: NONE

The vertical equilibrium option (keyword VE) should not be used with the miscible gas flood option.

Example

Using ECLIPSE 100:

MISCIBLE 2 25 TWOPOINT /

Keywords
Use dependence on surface tensions in properties (ECLIPSE 300)

The keyword does not require items 1 to 3 above. Any values are skipped. The keyword should be followed by a single slash (/).

If this option is used, parachor values should be entered in the PROPS section using the PARACHOR keyword, or the STVP keyword added to provide the variation of surface tension with pressure.

Example

Using ECLIPSE 300:

```
MISCIBLE
/
```
Miscibility region numbers

The keyword should be followed by one integer for every grid block in the current input box specifying the miscibility region to which it belongs.

**ECLIPSE 100 miscible flood model**

The region number should not be less than 0 or greater than NTMISC (see keyword **MISCIBLE**). The data must be terminated by a slash (/).

A positive value of the miscibility region number specifies which value of the mixing parameter (input using the keyword **TLMIXPAR**) should be used to calculate the effective PVT properties of the gas and oil components in each cell of the grid. A positive miscibility region number also specifies which table of miscible residual oil saturation versus water saturation should be used in the calculation of the relative permeabilities and effective oil and gas PVT properties.

Grid blocks which are allocated a miscibility region number of 0 are assumed to contain immiscible oil and gas. The mixing parameter is set automatically to 0 in these regions and the gas/oil relative permeabilities are computed from immiscible data. Any cells for which **MISCNUM** is not explicitly defined are given a **MISCNUM** value of 1.

The miscibility region numbers can be modified during the simulation by re-specifying the **MISCNUM** keyword in the **SCHEDULE** section. This procedure should only be carried out with care and is **not** recommended in general.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

The **MISCNUM** keyword may only be used in miscible flood simulations (keyword **MISCIBLE** in **RUNSPEC**). However, the **MISCNUM** keyword is not obligatory in these cases - a default region number of 1 is assumed for each grid cell. When using the polymer flood model (keyword **POLYMER** in **RUNSPEC**) the **PLMIXNUM** keyword should be used instead.

See also the keywords **SORWMIS** and **TLMIXPAR** in the **PROPS** section.

The ECLIPSE 100 treatment of miscibility is described in "Miscible flood modeling" in the **ECLIPSE Technical Description**.

**Example**

```plaintext
-- In the RUNSPEC section:
MISCIBLE
2 /
DIMENS
8 6 5 /
-- In the REGIONS section (with no input box set):
MISCNUM
72*0 72*1 96*2 /
```

**ECLIPSE 300 miscible model - surface tension effects**

When using the **MISCIBLE** option, the default treatment is to allow the relative permeabilities to approach straight lines as the interphase surface tension approaches zero. This is described in "Saturation functions" in the **ECLIPSE Technical Description**. **MISCNUM** allows the user to specify a saturation table to be used in this limit. This option allows the user to use a saturation table to control numerical dispersion at low surface
tensions. This option should be used with care in the near critical region, as there may be a discontinuity if the system becomes single phase.

The saturation function region number specifies which set of saturation functions (input using `SGFN`, `SOF2` and related keywords) should be used to calculate relative permeabilities and capillary pressures in each grid block in the miscible limit.

**Note:** In ECLIPSE 300 the `MISCNUM` keyword is not permitted with end point scaling. If this is set with `ENDSCALE`, `MISCNUM` will use UNSCALED endpoints.

### Example

With 240 cells and two miscible limit regions:

```
MISCNUM
  144*1  96*2  /
```
Miscibility surface tension reference

When the MISCIBLE option is used, this keyword allows the reference surface tension to be set. If MISCSTR is not used, the average initial state surface tension is used as the reference. The keyword is followed by some or all of these data items, terminated with a slash:

1. The reference surface tension - flow is immiscible when the surface tension is greater than or equal to this value.

   UNITS: dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT-M)

2. The maximum surface tension expected.

   UNITS: dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT-M)

   If a non-default value is entered, a warning is issued after each report step if a surface tension has been calculated that exceeds this maximum. Note that during the relative permeability interpolation the curves are taken to be the immiscible curves for cells with a surface tensions above the reference value.

   Values should not be less than the reference surface tension - if they are they will be increased to equal the reference surface tension.

3. The maximum surface tension used to scale the input capillary pressure curves.

   UNITS: dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT-M)

   If this data item is not set, the maximum assumed is the reference value. Setting item 3 allows scaling of the capillary pressure beyond the reference value of surface tension.

   Values should not be less than the reference surface tension - if they are they will be increased to equal the reference surface tension.

The use of these parameters is discussed in "Surface tension effects" in the ECLIPSE Technical Description.

See also keyword MISCSTRP.

Example

[MISCSTR 52 /]
When the MISCIBLE option is used, this keyword is an alternative to MISCSTR for specifying the surface tension at which immiscible conditions are deemed to exist.

The problem with MISCSTR is that you will not generally know the surface tension at which the rock curve data was measured. MISCSTRP allows a pressure to be specified at which the reservoir fluid composition will be flashed, to obtain the reference value. This will usually be well below the initial reservoir pressure, far enough away from the critical point for the oil and gas to be thoroughly immiscible.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT–M)
MISCSTRR  Miscibility reference surface tension by saturation region

When the **MISCIBLE** option is used, this keyword allows the reference surface tension to be set by saturation region number. If no reference surface tension is set or implied via any of the keywords **MISCSTR, MISCSTRR, MISCSTRP** or **STVP** then the average initial surface tension, calculated using the Macleod Sugden correlation, is used as the reference for the whole reservoir.

For each saturation table present, the keyword is followed by some or all of these data items, terminated with a slash:

1. The user reference surface tension - flow is immiscible when the surface tension is greater than or equal to this value.
   
   **UNITS:** dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT–M)
   
   If this value is defaulted, then the average value of the initial surface tension over this saturation region is used.

2. The maximum surface tension expected.
   
   **UNITS:** dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT–M)
   
   If a non-default value is entered, a warning is issued after each report step if a surface tension has been calculated that exceeds this maximum, for this saturation region.

   Values should not be less than the reference surface tension - if they are they will be increased to equal the reference surface tension.

3. The maximum surface tension used to scale the input capillary pressure curves.
   
   **UNITS:** dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT–M)
   
   If this data item is not set, the maximum assumed is the reference value. Setting item 3 allows scaling of the capillary pressure beyond the reference value of surface tension, for this saturation region.

   Values should not be less than the reference surface tension - if they are they will be increased to equal the reference surface tension.

4. The pressure at which the reservoir fluid composition is flashed, to obtain the reference value. This item overrides any value placed in item 1.
   
   **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT–M)
   
   The problem with using item 1 is that the you do not generally know the surface tension at which the rock curve data was measured. This pressure is usually well below the initial reservoir pressure, far enough away from the critical point for the oil and gas to be thoroughly immiscible.

The use of these parameters is described in "Surface tension effects" in the **ECLIPSE Technical Description**.

The **MISCSTRR** keyword overrides any data present in the **MISCSTR** and **MISCSTRP** keywords. However, the **STVP** keyword overrides data in the **MISCSTRR** keyword.

**Example**

```
MISCSTRR
1* 1* 1* 10 /
```
<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>3.2</td>
<td>3.9</td>
<td>1*</td>
<td>/</td>
</tr>
<tr>
<td>1*</td>
<td>3.6</td>
<td>1*</td>
<td>1*</td>
</tr>
</tbody>
</table>

**Keywords**

1335
MIXACTG Specify gas activity functions for aqueous components

This keyword must be used to specify the gas activity function, for each aqueous component, when the MIXWPERM option is activated. The data for each soluble component consists of two records. The first record is the component index. This is followed by a table consisting of four columns of data. Each record is terminated by a slash. A null record, that is, a record with no data before the terminating slash, must end this keyword.

This keyword is only compatible with the CO2SOL, H2SSOL and GASSOL options and must be entered together with MIXWPERM and MIXACTO. Note that, when used in conjunction with GASSOL, keyword MIXACTG must be entered after the SOLUAQA keyword.

This keyword must only be used in FULLIMP mode.

For further details concerning the associated mixed water permeability model, see "Mixed water relative permeability modeling" in the ECLIPSE Technical Description.

Record 1

One item of data:

1. The component index defined by the CNAMEs keyword. Note that this index should correspond to CO2 if the CO2SOL option is being used and H2S if the H2SSOL option is being used. When the GASSOL option is used, the set of indices entered should match those listed, in a similar fashion, under the SOLUAQA keyword.

Record 2

A table consisting of four columns of data.

1. Pressure.
   Values should increase monotonically down the column.
   UNITS: barsa (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)

2. Activity coefficient of $T^0$ in the gas activity function, at the specified pressure.
   UNITS: Dimensionless

3. Activity coefficient of $T$ in the gas activity function, at the specified pressure.
   UNITS: 1/K (METRIC), 1/°R (FIELD), 1/K (LAB), 1/K (PVT-M)

4. Activity coefficient of $T^2$ in the gas activity function, at the specified pressure.
   UNITS: 1/K² (METRIC), 1/°R² (FIELD), 1/K² (LAB), 1/K² (PVT-M)

Example

```
MIXACTG
2/
-- Pressure     bi1    bi2    bi3
   1.00000  0.0001  0.0001  0.0001
   100.0000  0.0001  0.0001  0.0001
   200.0000  0.0001  0.0001  0.0001
```
<table>
<thead>
<tr>
<th>Pressure</th>
<th>b₁₁</th>
<th>b₁₂</th>
<th>b₁₃</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0000</td>
<td>0.0001</td>
<td>0.0001</td>
<td>0.0001</td>
</tr>
<tr>
<td>100.0000</td>
<td>0.0001</td>
<td>0.0001</td>
<td>0.0001</td>
</tr>
<tr>
<td>200.0000</td>
<td>0.0001</td>
<td>0.0001</td>
<td>0.0001</td>
</tr>
</tbody>
</table>

If the gas activity coefficients listed above are repeated for the oil activity coefficients (see keyword `MIXACTO`) then, for each pressure specified in the example above, the activity parameters are valid for all temperatures up to 70 degrees Celsius.
**MIXACTO** Specify oil activity functions for aqueous components

This keyword must be used to specify the oil activity function, for each aqueous component, when the MIXWPERM option is activated. The data for each soluble component consists of two records. The first record is the component index. This is followed by a table consisting of four columns of data. Each record is terminated by a slash. A null record, that is, a record with no data before the terminating slash, must end this keyword.

This keyword is only compatible with the CO2SOL, H2SSOL and GASSOL options and must be entered together with MIXWPERM and MIXACTG. Note that, when used in conjunction with GASSOL, keyword MIXACTO must be entered after the SOLUAQA keyword.

This keyword must only be used in **FULLIMP** mode.

For further details concerning the associated mixed water permeability model, see "Mixed water relative permeability modeling" in the *ECLIPSE Technical Description*.

**Record 1**

One item of data:

1. The component index defined by the **CNames** keyword. Note that this index should correspond to CO$_2$ if the **CO2SOL** option is being used and H$_2$S if the **H2SSOL** option is being used. When the **GASSOL** option is used, the set of indices entered should match those listed, in a similar fashion, under the **SOLUAQA** keyword.

**Record 2**

A table consisting of four columns of data.

1. Pressure.
   
   Values should increase monotonically down the column.
   
   **UNITS:** barsa (**METRIC**), psia (**FIELD**), atm (**LAB**), atm (**PVT-M**)

2. Activity coefficient of $T^0$ in the oil activity function, at the specified pressure.
   
   **UNITS:** Dimensionless

3. Activity coefficient of $T$ in the oil activity function, at the specified pressure.
   
   **UNITS:** 1/K (**METRIC**), 1/°R (**FIELD**), 1/K (**LAB**), 1/K (**PVT-M**)

4. Activity coefficient of $T^2$ in the oil activity function, at the specified pressure.
   
   **UNITS:** 1/K$^2$ (**METRIC**), 1/°R$^2$ (**FIELD**), 1/K$^2$ (**LAB**), 1/K$^2$ (**PVT-M**)

**Example**

```
MIXACTO
  2/
  --Pressure     a1   a12  a13
  1.00000  0.0001  0.0001  0.0001
  100.0000  0.0001  0.0001  0.0001
  200.0000  0.0001  0.0001  0.0001
```
If the oil activity coefficients listed above are repeated for the gas activity coefficients (see keyword `MIXACTG`) then, for each pressure specified in the example above, the activity parameters are valid for all temperatures up to 70 degrees Celsius.
**MIXWPERM**  
**Mixed water permeability option**

This keyword activates a modified water relative permeability calculation which takes into account the dissolution of gases. It must be used in conjunction with keywords `MIXACTO` and `MIXACTG` and is only compatible with the `CO2SOL`, `H2SSOL` and `GASSOL` options.

The `MIXWPERM` keyword has no associated data and must only be used in `FULLIMP` mode.

For further details concerning this alternative model, see "Mixed water relative permeability modeling" in the *ECLIPSE Technical Description*.

**Example**

```plaintext
MIXWPERM
```

---

**Keywords**

```plaintext
ECLIPSE 100
x ECLIPSE 300
x SPECIAL
RUNSPEC
GRID
EDIT
x PROPS
REGION
SOLUTION
SUMMARY
SCHEDULE
```
MLANG

**Maximum surface gas concentration**

This keyword is optional, and is used to scale the LANGMUIR Isotherm data on a cell-by-cell basis. See "Coal bed methane model" in the *ECLIPSE Technical Description* for further information.

The keyword should be followed by one real number for each cell in the current input box. The values denote the maximum surface gas concentration in the scaled LANGMUIR table. Data need only be supplied for the matrix cells, that is the upper half of the grid. The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*).

**UNITS:** sm$^3$/m$^3$ (*METRIC*), Mscl/ft$^3$ (*FIELD*), scc/cc (*LAB*)

The keyword can be used in conjunction with the EQUALS, ADD and MULTIPLY keywords.

See also keywords LANGMUIR, MLANGLSV and GASSATC.

**Example**

```plaintext
--- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX 1 4 1 4 1 1 /
MLANG
78.0 77.0 64.1 31.0
77.0 70.0 63.2 25.1
63.0 63.4 58.4 21.4
44.5 31.6 28.4 8.2 /
```
MLANGSLV  Maximum solvent surface concentration

This keyword is optional and is used to scale the LANGSOLV isotherm data on a cell-by-cell basis. See "Coal bed methane model" in the ECLIPSE Technical Description for further information.

The keyword should be followed by one real number for each cell in the current input box. The values denote the maximum solvent surface concentration in the scaled LANGSOLV table. Data need only be supplied for the matrix cells, that is the upper half of the grid. The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*).

UNITs: sm$^3$/m$^3$ (METRIC), Mscf/ft$^3$ (FIELD), scc/cc (LAB)

The keyword can be used in conjunction with the EQUALS, ADD and MULTIPLY keywords.

See also keywords LANGMUIR and MLANG.

Example

```
---------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX       1   8   1   8   1 /
MLANGSLV  8*64.0
          8*62.7
          8*61.0
          8*60.3
          8*59.1
          8*57.1
          8*53.0
          8*34.0 /
```
MONITOR

Request output for run-time monitoring

This keyword requests that information required by GRAF for the run-time monitoring option should be written to the Summary Specification file.

By default, runtime monitoring is always on.

The MONITOR keyword has no associated data.

See also keyword NOMONITO.

Example

```
MONITOR
```

Keywords

1343
Specify options for tensor permeabilities

This keyword specifies that multipoint transmissibilities are to be used in the discretized flow equations. If tensor permeabilities are entered then the discretization accounts for the tensor permeabilities in a consistent manner.

The keyword should be followed by two items followed by a slash (/).

1. Specify the basis in which the permeability tensor is defined.

   The permeability tensor $K$ is assumed to be symmetric and is specified by its physical components $K_{11} = \text{PERMX}$, $K_{22} = \text{PERMY}$, $K_{33} = \text{PERMZ}$, $K_{12} = \text{PERMXY}$, $K_{23} = \text{PERMYZ}$ and $K_{31} = \text{PERMZX}$.

   The tensor may be supplied in the following bases:

   1. $K$ is defined in the Cartesian x, y and z axes.
   2. $K$ is defined in each corner point cell relative to the coordinate system formed by joining the midpoints of opposite faces in a corner point cell.
   3. $K$ is defined in the bedding plane by the vectors obtained by joining the midpoints of opposite faces in the corner point cell in the logical $i$ and $j$ directions. The third basis vector is chosen normal to the bedding plane.

   DEFAULT: 2

2. Type of transmissibility calculation used in the multipoint flux approximation.

   0. Ignore all the cross terms and calculate standard two-point corner point transmissibilities. This diagonalizes the permeability tensor in a form suitable for the standard \texttt{NEWTRAN} calculation. In the absence of faults, it is equivalent to the standard \texttt{HALFTRAN} calculation when the vector $K \vec{d}$ is parallel to the common cell face normal in the half of each connecting cell adjacent to the common face, where $\vec{d}$ is the vector joining either cell center to the center of the common face.

   1. The ‘O’ method is used to give a 27 point stencil in 3D or a 9 point stencil in 2D.
   2. The ‘U’ method is used to give a 19 point stencil in 3D or a 9 point stencil in 2D.

   DEFAULT: 0

\textbf{Note:} The multipoint flux approximation deals with both non-orthogonality and the tensor permeability. For an orthogonal grid with the principal axes of the tensor aligned with the coordinate directions, the standard 5 or 7 point schemes are recovered.

See "Tensor permeability" in the \textit{ECLIPSE Technical Description} for further information.

\textbf{Note:} The use of the MPFA option is not allowed with the \texttt{WARP} solver. Please use the pre-2002A solver (ECLSOL) instead by setting item 66 in \texttt{OPTIONS3} to a non-zero value.
Figure 3.10. Stencil for nine point MPFA scheme in 2D

Figure 3.11. Stencil for ‘O’
Figure 3.12. Stencil for ‘U’

Example

<table>
<thead>
<tr>
<th>MPFA</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 1 /</td>
</tr>
</tbody>
</table>
**MPFANUM**  Identifies regions where the multipoint flux discretization is used

The **MPFANUM** keyword indicates regions in which to apply the multipoint flux discretization (see keywords **MPFA** and **MPFNNC** and also “Tensor permeability” in the **ECLIPSE Technical Description**).

The **MPFANUM** keyword should be followed by one integer for every grid block in the current input box, specifying the discretization flag for each grid block. The flag should be set either to 1 to indicate that the multipoint flux discretization is to be employed, or to zero in regions where the standard finite-difference discretization is required (five points in 2D, seven points in 3D).

Any grid blocks not explicitly assigned a **MPFANUM** value are defaulted to a value of 1. You must set the **MPFANUM** value explicitly to zero for any regions where the nine-point scheme is not required.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

When using the ECLIPSE 100 Local Grid Refinement option, the **MPFANUM** array should not be defined between the **CARFIN** and **ENDFIN** keywords. Within local grids, the **MPFANUM** array is inherited from the host cells in the global grid.

**Example**

To apply the **MPFA** scheme only in a 2*2*2 region in a 4*4*3 field:

```
MPFANUM
  1 1 0 0
  1 1 0 0
  0 0 0 0
  0 0 0 0
  1 1 0 0
  1 1 0 0
  0 0 0 0
  0 0 0 0
16*0 /
```

---

**Keywords**

- **ECLIPSE 100**
- **ECLIPSE 300**
- **SPECIAL**
- **RUNSPEC**
- **GRID**
- **PROPS**
- **REGIONS**
- **SOLUTION**
- **SUMMARY**
- **SCHEDULE**

---

1347
Explicit entry of multi-point flux non-neighbor connections

This keyword specifies transmissibilities for multipoint flows.

The form of the keyword to define a single multipoint NNC is as follows:

```
MPFNNC
IX IY IZ JX JY JZ TRANP /
KX KY KZ TRANS /
.............. /
/
/
```

Each line defining the primary non-neighbor connection (NNC) is terminated with a slash (/). This is followed by any number of lines defining the associated secondary NNCs. End the list of secondary NNCs with a single slash (/). After the last non-neighbor connection definition, a single slash terminates the list. It is possible to specify this keyword more than once in the grid data.

The items for each multipoint NNC are:

- **IX**, **IY**, **IZ** The coordinates of the first cell joined to the primary non-neighbor connection.
- **JX**, **JY**, **JZ** The coordinates of the second cell joined to the primary non-neighbor connection.
- **TRANP** The transmissibility of the primary non-neighbor connection

**UNITS:** cP.rm$^3$/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm$^3$/day/atm (PVT-M)

- **KX**, **KY**, **KZ** The coordinates of the second cell joined to **IX**, **IY**, **IZ** by the secondary non-neighbor connection
- **TRANS** The transmissibility of the secondary non-neighbor connection

**UNITS:** cP.rm$^3$/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm$^3$/day/atm (PVT-M)

**Note:** A non-neighbor connection is allowed between two cells that are actually neighbors (or between two cells where ECLIPSE would form a non-neighbor connection in the absence of the MPFNNC keyword); this simply adds to the existing transmissibility. Any multipoint NNC input with transmissibility less than 0.000001 results in a warning and is ignored. Connections to inactive cells result in a warning, and are ignored.

**Note:** All the transmissibilities need to be generated externally and the MPFNNC keyword should be used to apply the multipoint flux capability within ECLIPSE 100. This facility is intended primarily for use with unstructured grids, and the multi-point NNC coefficients could be provided by FloGrid. On structured grids, tensor permeabilities may be used in ECLIPSE 300 and the MPFA keyword activates the multipoint flux scheme. The MPFNNC keyword should not be used in the same data set as the MPFA keyword.
Note: The multipoint flux facility is a rigorous treatment to avoid discretization errors arising from grid non-orthogonality. However, the large number of connections and extra calculations required in the simulator mean that the run time will increase considerably.

Example

```
MPFMNC
11 11 11 10 10 10 1.0 /
 22 22 22 2.0 /
 33 33 33 3.0 /
 44 44 44 4.0 /
/
23 24 25 11 12 13 7.0 /
 32 32 32 1.0 /
 33 33 33 2.0 /
 34 34 34 3.0 /
/
```
**Miscible saturation functions**

This keyword is entirely optional and should only be used in miscible runs (keyword `MISCIBLE` in the `RUNSPEC` section) and in runs using the Solvent Model. It allows the use of a relative permeability curve for the miscible displacement. See "Miscible flood modeling" and "Solvent model" in the `ECLIPSE Technical Description` for further details of these options.

The keyword data comprises `NTSFUN` (see keyword `TABDIMS`) tables of solvent/oil miscible saturation functions, each terminated by a slash `/`.

Each table consists of 3 columns of data. The first column is the normalized total gas fraction (that is solvent and gas saturation if Solvent and only gas saturation if no Solvent). The next two columns are the relative permeability multipliers for the (gas+solvent) and the oil respectively ($M_{krsg}$ and $M_{kro}$).

1. The normalized total gas phase fraction (gas+solvent)
   - Values should span the range 0 to 1 and increase monotonically down the column.

2. The corresponding (gas+solvent) relative permeability multiplier.
   - Values should be between 0 and 1 and should be level or increasing down the column.

3. The corresponding oil relative permeability multiplier.
   - Values should be between 0 and 1 and should be level or decreasing down the column.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than `NSSFUN` (see keyword `TABDIMS`).

Note that the relative permeabilities supplied in this keyword are normalized and honor the end-point supplied in the `SORWMIS` and `SGCWMIS` keywords. `SORWMIS` supplies the critical oil saturation as a function of water saturation, and `SGCWMIS` the critical gas saturation. Hence when the flood is totally miscible (see the `MISC` and `PMISC` keywords) the (gas+solvent) and the oil relative permeabilities are taken as:

\[
\begin{align*}
    k_{rsg} &= M_{krsg} \left( \frac{S_g + S_s - S_{gc}}{S_n - S_{gc} - S_{or}} \right) k_m (S_n) \quad \text{Eq. 3.118} \\
    k_{ro} &= M_{kro} \left( \frac{S_o - S_{or}}{S_n - S_{gc} - S_{or}} \right) k_m (S_n) \quad \text{Eq. 3.119}
\end{align*}
\]

If no solvent is present then $S_s = 0$.

Where

- $S_g$ is the gas saturation
- $S_s$ is the solvent saturation (which is zero if the solvent model is not used)
- $S_n$ is the total hydrocarbon saturation $S_n = S_o + S_g + S_s$
- $S_{gc}$ is the critical gas saturation, `SGCWMIS`
- $S_{or}$ is the critical oil saturation, `SORWMIS`
$k_{rn}(S_n)$ is the relative permeability of hydrocarbon to water.

If the MSFN keyword is not supplied, the relative permeabilities are taken as straight line functions, that is

$$k_{rs} = \frac{S_g + S_s - S_{gc}}{S_n - S_{gc} - S_{or}} k_{rn}(S_n) \quad \text{Eq. 3.120}$$

$$k_{ro} = \frac{S_o - S_{or}}{S_n - S_{gc} - S_{or}} k_{rn}(S_n) \quad \text{Eq. 3.121}$$

See also the MISC, PMISC, SSFN, SORWMIS and SGCWMIS keywords.

**Example**

With NTSFUN=2 and NSSFUN≥6

<table>
<thead>
<tr>
<th>Sfrac</th>
<th>MKrs</th>
<th>MKro</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.0</td>
<td>1.0</td>
</tr>
<tr>
<td>1.0</td>
<td>1.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Sfrac</th>
<th>MKrs</th>
<th>MKro</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.0</td>
<td>1.0</td>
</tr>
<tr>
<td>0.4</td>
<td>0.2</td>
<td>0.5</td>
</tr>
<tr>
<td>0.6</td>
<td>0.4</td>
<td>0.4</td>
</tr>
<tr>
<td>1.0</td>
<td>1.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>
Control output of the message file

The keyword should be followed by an integer.

The possible values are:

1. Control of the MSG file output
   - 0 A MSG file is produced
   - 1 A MSG file is produced.

   DEFAULT: 1

The data must be terminated by a slash (/).

If the MSGFILE keyword is not used, then by default, a MSG file is produced.

The message file feature is provided primarily for post-processor convenience, and we recommend that you consult the print file for the full context of messages.

Example

```plaintext
MSGFILE
0 /
```
**MULSGGD**  
**Dual porosity matrix-fracture coupling multiplier for oil-gas gravity drainage**

If the dual porosity option is being used (keyword `DUALPORO` in the `RUNSPEC` section), and the alternative matrix-fracture coupling transmissibilities for oil-gas gravity drainage have been specified in the grid section through the `SIGMAGD` or `SIGMAGDV` keywords, then this keyword can be used to specify a multiplier in the `SCHEDULE` section which will modify further these matrix-fracture coupling transmissibilities.

If this keyword is used, a single multiplier is applied to the entire current grid. To specify values for individual cells, use the `MULSGGDEV` keyword.

The keyword should be followed by one single positive real number, which will be applied to every cell in the current grid.

**UNITS:** dimensionless

**Example**

```
MULSGGD
0.5  /
```
MULSGGDV  Dual porosity matrix-fracture coupling multiplier for oil-gas gravity drainage

If the dual porosity option is being used (keyword DUALPORO in the RUNSPEC section), and the alternative matrix-fracture coupling transmissibilities for oil-gas gravity drainage have been specified in the grid section through the SIGMAGD or SIGMAGDV keywords, then this keyword can be used to specify a multiplier in the SCHEDULE section which will modify further these matrix-fracture coupling transmissibilities.

The keyword should be followed by one positive real number for every grid block in the current box within the top NDIVIZ/2 layers of the grid (that is, the matrix cells). Any values input in the bottom half of the grid (the lower NDIVIZ/2 layers) will be ignored.

To specify a single value for all cells, use the MULSGGD keyword.

UNITS: dimensionless

Example

```plaintext
------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 8 1 1 1 2 /
MULSGGDV
0.5 0.5 0.5 0.5 0.5 0.5
0.3 0.3 0.3 0.3 0.3 0.3 /
```

Keywords
MULTFLT

**Modifies the transmissibility across a named fault**

The MULTFLT keyword can be used to modify the transmissibility (and diffusivity) across a fault previously defined using the FAULTS keyword.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/):

1. Fault name (up to 8 characters)
   (as specified in the FAULTS keyword)
   A fault root name, ending with an asterisk (*) can also be used to refer to several faults in one record.
2. Transmissibility multiplier
   DEFAULT: 1.0
3. Diffusivity multiplier
   (used only if the Diffusion option is enabled with the DIFFUSE keyword in the RUNSPEC section)
   DEFAULT: 1.0

The set of records must end with a blank record containing only a slash (/).

**Notes**

- If a fault is referred to twice in one or more MULTFLT keywords, the transmissibility multiplier is taken from the last entry.
- If the fault has been specified with a face X, Y or Z in the FAULTS keyword, then using the MULTFLT keyword in the GRID section modifies the MULTX, MULTY and MULTZ arrays along the fault trajectory. If the MULTX, MULTY or MULTZ keyword is used for some cells, in addition to the MULTFLT keyword, then the resultant transmissibility multiplier is the product of the two input multipliers.

Similarly, if negative direction transmissibility multipliers are enabled (using the GRIDOPTS keyword in the RUNSPEC section), and if the fault has been specified with a face X-, Y- or Z- in the FAULTS keyword, then using the MULTFLT keyword in the GRID section modifies the MULTX-, MULTY- and MULTZ- arrays along the fault trajectory. If the MULTX-, MULTY- or MULTZ- keyword is used for some cells, in addition to the MULTFLT keyword, then the resultant transmissibility multiplier is the product of the two input multipliers.

- Use of the MULTFLT keyword in the EDIT section multiplies the transmissibilities along fault trajectories specified by the FAULTS keyword in the GRID section. The transmissibilities for any non-neighbor connections across such a fault are also modified to reflect the multiplier analogously to the use of MULTX, MULTY or MULTZ.

Similarly, with the MULTX-, MULTY-, MULTZ- keywords.

- If two faults with different names run along the same face of a cell, then both transmissibility multipliers are applied.
- Refer also to the keywords, FAULTS and RPTGRID.
- The MULTFLT keyword can be used in the SCHEDULE section to modify the transmissibilities along the fault trajectory during the run. The effect of this keyword in the SCHEDULE section is cumulative.
so every time the MULTFLT keyword is encountered, it multiplies the current transmissibility across a specific face.

- The modified transmissibilities can be output to the Print file by using the mnemonic MULT in the RPTSCHED keyword.

- If the transmissibilities have been imported from a GSG file rather than calculated (by using the TRANPORV mnemonic in the PETOPTS keyword) MULTFLT will be ignored in the GRID section and must be specified in the EDIT section instead if required.

- If a fault is specified twice, the transmissibility multiplier will be applied twice. In ECLIPSE 300, this behavior can be turned off by setting item 248 of the OPTIONS3 keyword to 1.

- If TRANX, TRANY or TRANZ keywords are used then MULTFLT should appear after these keywords in the EDIT section. Otherwise, as TRANX, TRANY and TRANZ set transmissibility explicitly, they will selectively override MULTFLT for neighboring connections.

**Example**

Refer to the FAULTS keyword example:

```
MULTFLT
   -- Multiplier (no diffusion)
   zigzag 0.2 /
   block  1.4 /
```

Keywords
MULTIN

Indicates that input files are multiple

This indicates that input files, which may be either multiple or unified, are to be multiple. This is the default; unified files may be requested with the UNIFIN keyword.

Multiple files enable the progress of the run to be monitored, and are generally preferred; but they do produce a larger number of files, which may be inconvenient on machines without a directory structure.

This keyword has no associated data.
MULTIPHA  Perform multi-phase flash

This keyword requests the multiphase flash which can be used to predict asphaltene precipitation. See "Multi-phase flash" in the ECLIPSE Technical Description. The results of this flash are used for reporting only, and not for simulation. For simulation see the "Asphaltene option" in the ECLIPSE Technical Description.

The keyword cannot be used with Blackoil or Thermal runs. It has no associated data.

Example

```
MULTIPHA
```

Keywords

1358
MULTIPLY

Multiply array by a constant in current box

The keyword may be followed by any number of records, each of which is terminated by a slash (/). The
data is terminated by a null record, with no data before the terminating slash(/).

Each record consists of at least two items of data:

1. The name of the array to be modified
2. The constant by which the array, specified by item 1, is to be multiplied
   The constant should not be negative, but may be real or integer. This item should not be defaulted.

   Note: Negative multipliers and more complex arithmetic operations are available using the OPERATE
   keyword.

The following items of data define the limits of a box within the grid over which the value is to be set.
These items may be used to redefine the input box for this and subsequent operations within the
current keyword. The values are used until reset or until the end of the keyword.
If they are not defined (a slash is inserted after item 2), they default to the values which were used for
the previous operation within the current keyword.
For the first operation in the keyword, the box defaults to the values set by the most recent BOX or
ENDBOX keyword. If there is no preceding BOX or ENDBOX in the current section, the box is taken to
include the entire reservoir.

3. First block to be modified on the X axis (IX1)
4. Last block to be modified on the X axis (IX2)
5. First block to be modified on the Y axis (JY1)
6. Last block to be modified on the Y axis (JY2)
7. First block to be modified on the Z axis (KZ1)
8. Last block to be modified on the Z axis (KZ2)

The data must satisfy:
1 ≤ IX1 ≤ IX2 ≤ NDX
1 ≤ JY1 ≤ JY2 ≤ NDY
1 ≤ KZ1 ≤ KZ2 ≤ NDZ,
where NDX, NDY, NDZ are the limits in the current BOX.

ECLIPSE 100
If the MULTIPLY keyword is used in the PROPS section to modify the end-point scaling arrays, the
modification is only made to the end points that have been previously defined. If a cell has not been
assigned a value, the end point defaults to the table value.
MULTIPLY is now available in the REGIONS and SOLUTION sections.

ECLIPSE 100
TNUM arrays can also be used with the MULTIPLY keyword.

ECLIPSE 100
In the REGIONS section only integer values should be used.
Legal arrays

GRID section arrays

- DX (DR)
- DY (DTHETA)
- DZ
- PERMX (PERMR)
- PERMY (PERMTHT)
- PERMZ
- MULTX (MULTR)
- MLYT (MULTHT)
- MULTZ
- PORO
- NTG
- DZNET
- TOPS
- DIFFMX (DIFFMR)
- DIFFMY (DIFFMTHT)
- DIFFMZ

ECLIPSE 300 only

- FLUXNUM
- MULTNUM
- MPFANUM

EDIT section arrays

- PORV
- DEPTH
- TRANX (TRANR)
- TRANY (TRANTHT)
- TRANZ
- DIFFX (DIFFR)
- DIFFY (DIFFHT)
- DIFFZ
- HEATTX (HEATTR)
- HEATTY (HEATTHT)
- HEATTZ (HEATTZ)
- ROCKV

PROPS section arrays

- SWL
- SWCR
- SWU
- SGL
- SGCR
- SGU
- KRW
- KRO
- KRG
- PCG
- PCW

REGIONS section arrays

- SATNUM
- PVNUM
- EQLNUM
- IMBNUM
- FIPNUM
- ENDNUM
- ROCKNUM
- MISCNUM
- WH2NUM
- WH3NUM
- WH3NUM
- EOSNUM

ECLIPSE 300 only

See also keywords BOX, ENDBOX, EQUALS, ADD and COPY.
SOLUTION section arrays

PRESSURE  SWAT  SGAS
RV        RS     TBLK

**ECLIPSE 100 only**
GI         OILAPI  SALT

**ECLIPSE 100 only**
GASCONC    SOLVCONC  SOLVFRAC

**ECLIPSE 100 only**
SSOL       SURF     SFOAM

**ECLIPSE 300 only**
SPOLY

**ECLIPSE 300 only**
SOIL       PBUB     PDEW

**ECLIPSE 300 only**
TEMP       TEMPI

Notes

- The list above is not exhaustive, but includes the most common examples.
- Keywords in brackets are for radial geometry runs.
- The multiplied quantity may be less than unity, in order to divide.
- The use of the MULTIPLY keyword assumes that a value has already been assigned to the array, either explicitly or using the EQUALS keyword (except in the EDIT section, where the array has already been calculated after processing the GRID section).
- The ADD, COPY, EQUALS and MULTIPLY keywords are processed as they are read, so that repeated operations are possible.
- **Item 2 should not be defaulted.** However, this facility is retained for back-compatibility. If this item is defaulted, this will be treated differently by ECLIPSE 100 and ECLIPSE 300. For ECLIPSE 100, the defaulted value is assumed to be undefined, whereas for ECLIPSE 300, the defaulted value is assumed to be zero. The subsequent processing will depend, therefore, both upon the specific keyword to which the operation is being applied and which simulator is being used.

Examples

Example 1

In the EDIT section:

```
------------- ARRAY   CONSTANT ----- BOX ----- 
MULTIPLY
TRANS   500        3 8 1 10 3 3  /
TRANSZ  0          / defaults to last specified box
PORV    100.0      / defaults to last specified box
```

Example 2

In the GRID section:

```
------------- ARRAY   CONSTANT ----- BOX ----- 
MULTIPLY
```
<table>
<thead>
<tr>
<th>Keyword</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PERMX</td>
<td>0.1</td>
<td>3 8 1 10 3 3 /</td>
</tr>
<tr>
<td>PERMZ</td>
<td>0</td>
<td>/ defaults to last specified box</td>
</tr>
<tr>
<td>PORO</td>
<td>0.2</td>
<td>/ defaults to last specified box</td>
</tr>
</tbody>
</table>

Keywords
MULTIREG Multiplies an array by a constant in a given region

MULTIREG is similar to the MULTIPLY keyword, but uses either FLUXNUM flux regions, MULTNUM regions or OPERNUM regions, rather than boxes of cells.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated by a slash (/).

The records can be terminated early using the slash character (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

Each record consists of the following items of data:

1. The name of the array to be modified
2. The constant by which the array, specified by item 1, is to be multiplied
   The constant should not be negative, and, for the REGIONS section, should be an integer.
   DEFAULT: 0.0
3. The flux region number, MULTNUM region number or OPERNUM region number (with choice of region determined using item 4) The flux region number refers to the region defined using the FLUXNUM keyword in the GRID section. It is not necessary to use the Flux Boundary option.
   The MULTNUM region number refers to the region defined using the MULTNUM keyword in the GRID section.
   The OPERNUM region number refers to the region defined using the OPERNUM keyword in the GRID or REGIONS section.
4. Choice of region: MULTNUM (M), FLUXNUM (F) or OPERNUM (O).
   DEFAULT: M

The set of records is ended by a blank record, containing only a slash (/).

Note: The FLUXNUM keyword, MULTNUM keyword or OPERNUM keyword must be entered before the MULTIREG keyword. If item 4 is defaulted, then the MULTNUM keyword is used.

In ECLIPSE 100, this keyword may not be used to modify the PBUB or PDEW arrays.
In ECLIPSE 300, this keyword may be used in SCHEDULE section. However, this is a specialist facility.

Note: In the SCHEDULE section, regions (predefined by the FIPxxxx keyword in the REGIONS section) can be used in item 4.

Legal arrays

GRID section arrays

<table>
<thead>
<tr>
<th>DX (DR)</th>
<th>DY (DTHETA)</th>
<th>DZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>PERMX (PERMR)</td>
<td>PERMY (PERMTHT)</td>
<td>PERMZ</td>
</tr>
</tbody>
</table>
**ECLIPSE Reference Manual**

MULTX (MULTR)  MULTY (MULTTHT)  MULTZ
PORO  NTG  DZNET
TOPS
DIFFMX (DIFFMR)  DIFFMY (DIFFMTHT)  DIFFMZ
FLUXNUM  MULTNUM  MPFANUM

**EDIT section arrays**

PORV  DEPTH
TRANX (TRANR)  TRANY (TRANHT)  TRANZ
DIFFX (DIFFR)  DIFFY (DIFFHT)  DIFFZ
HEATTX (HEATTR)  HEATTY (HEATTHT)  HEATTZ (HEATTZ)
ROCKV

**PROPS section arrays**

SWL  SWCR  SWU
SGL  SGCR  SGU
KRW  KRO  KRG
PCG  PCW

**REGIONS section arrays**

SATNUM  PVTNUM  EQLNUM
IMBNUM  FIPNUM  ENDNUM
ROCKNUM  MISCNUM  WH2NUM
WH3NUM

**SOLUTION section arrays**

PRESSURE  SWAT  SGAS
RV  RS  TBLK

**Keywords**

1364
ECLIPSE 100 only

- SSOL
- SPOLY
- SURF

ECLIPSE 100 only

- SFOAM

ECLIPSE 300 only

- SOIL
- PBUB
- PDEW

ECLIPSE 300 only

- TEMP

**SCHEDULE section arrays (specialist facility)**

ECLIPSE 300 only

- PORO
- PERMX
- PERMY

ECLIPSE 300 only

- PERM2
- MULTX
- MULTY

ECLIPSE 300 only

- MULTZ
- SIGMA(V)
- MULTSIG(V)

ECLIPSE 300 only

- MULSGGD(V)
- DIFFMMF
- WORK(1/2)

---

**Notes**

- The list above is not exhaustive, but includes the most common examples.
- Keywords in brackets are for radial geometry runs.
- The multiplied quantity may be less than unity, in order to divide.
- The use of the MULTIREG keyword assumes that a value has already been assigned to the array.
- Each record must refer to the same region. It is not possible to choose a different region for each data record.
- The MULTIREG keyword is not available for use with local grid systems (ECLIPSE 100 only).
- See also keywords EQUALREG, FLUXNUM, MULTNUM, OPERNUM, ADDREG, and COPYREG.

**Additional notes**

ECLIPSE 300 only

Additional notes for the use of this keyword in the SCHEDULE section. Again, this is a specialist facility. In the SCHEDULE section, the effect of this keyword will be as follows:

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DIFFMMF</td>
<td>Modifies the matrix-fracture diffusivity in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as DIFFMMFS.</td>
</tr>
<tr>
<td>MULSGGD</td>
<td>Modifies the matrix-fracture coupling transmissibility for oil-gas gravity drainage in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as MULSGGDS.</td>
</tr>
<tr>
<td>MULTSIG</td>
<td>Modifies the matrix-fracture coupling transmissibility in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as MULTSIGS.</td>
</tr>
<tr>
<td>MULTX</td>
<td>Modifies transmissibility in the X/Y/Z-direction. If output is required for flexible restart in the model, the multiplier values will be output as MULTXS/MULTYS/MULTZS.</td>
</tr>
</tbody>
</table>
### Keywords

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PERMX</td>
<td>Modifies permeability in the X/Y/Z-direction. Also approximates the corresponding transmissibility multiplier (\text{MULTX/MULTY/MULTZ}) determined from the ratio of the new permeability value to the old one. If output is required for flexible restart in the model, the permeability values will be output as (\text{MODPERMX/MODPERMY/MODPERMZ}).</td>
</tr>
<tr>
<td>PERMY</td>
<td></td>
</tr>
<tr>
<td>PERMZ</td>
<td></td>
</tr>
</tbody>
</table>

| PORO    | Modifies porosity. Modifies the corresponding pore volume by a multiplier \(\text{MULTPV}\) determined from the ratio of the new porosity value to the old one. If output is required for flexible restart in the model, the porosity values will be output as \(\text{MODPORO}\), and the pore volume multipliers will be output as \(\text{MULTPVS}\). |
| SIGMA   | Modifies the matrix-fracture coupling transmissibility in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as \(\text{SIGMA}\). |
| SIGMAV  | |

| WORK (1/2) | These are work arrays to be stored for subsequent use. If output is required for flexible restart in the model, these values will be output as \(\text{WORK (1/2)}\). |

### Table 3.20: Additional notes for MULTIREG in the SCHEDULE section

#### Examples

**Example 1**

Where \(\text{NX} = 4\), \(\text{NY} = 4\), \(\text{NZ} = 1\):

```
FLUXNUM
1  1  2  3
1  2  3  3
2  3  3  4
3  3  4  4
/
MULTIREG
PORO  1.1  1  F /
PERMX 2.0  2  F /
PERMY 2.0  2  F /
```

**Example 2**

\(\text{NX} = 6\), \(\text{NY} = 8\), \(\text{NZ} = 1\):

```
MULTNUM
1  1  2  2  2
1  1  2  2  2
4  4  1  3  3
4  4  1  3  3
4  4  1  3  3
5  5  5  5  5
4  4  1  3  3
4  4  1  3  3
/
MULTIREG
PORO  2.0  1 /
PORO  3.1  2 /
PERMX 2.1  4 /   
PERMX 2.1  3 /
PERMY 1.3  5 /
```
MULTMF Transmissibility multipliers for matrix-fracture coupling

This keyword specifies a transmissibility multiplier for flow between fractures and matrix in a dual porosity run (keyword DUALPOR in the RUNSPEC section).

The MULTMF keyword is followed by a value for each matrix cell, in normal grid order with the X-index changing fastest.
**MULTNUM**  Defines regions for applying inter-region transmissibility multipliers

This keyword provides an alternative region definition for applying inter-region multipliers, instead of applying them to flux regions defined by `FLUXNUM`. The `MULTNUM` keyword can be used to work with transmissibility multipliers using the keyword `MULTREGT`, or with the `MULTREGP` keyword (which sets a pore volume multiplier for a specific region).

To use this keyword, the maximum number of `MULTNUM` regions must first be defined. To do this, the 2nd item in the `GRIDOPTS` keyword in the `RUNSPEC` section must be set greater than zero.

The `MULTNUM` keyword should be followed by one integer for each grid cell, specifying the region number to which it belongs. The data must be terminated by a slash (/).

Any cells not explicitly assigned a value of `MULTNUM` will be taken to be in region 1.

**Note:** In ECLIPSE 100, 0 is a valid value for `MULTNUM` similarly to `FLUXNUM`. In ECLIPSE 300, 0 is an invalid value for `MULTNUM`.

In ECLIPSE 100, the `MULTNUM` array can be copied into the `REGIONS` section arrays (`FIPNUM` and `PVTNUM` for example) using the `COPY` keyword.

The `MULTNUM` array can be used with keywords `MULTIREG`, `ADDREG` and `EQUALREG` in the `GRID` section.

In ECLIPSE 100 only, the `MULTNUM` keyword cannot be used inside a `CARFIN/RADFIN...ENDFIN` bracket.

For use with the Dual Porosity option, only the matrix cells or the fracture cells need to be input in ECLIPSE 100. The cell values in the other porosity will be copied automatically. If the user wants to specify matrix cells and fracture cells separately, then item 117 in `OPTIONS` should be set greater than 0.

For use with Dual Porosity option, both matrix and fracture cells should be provided for ECLIPSE 300. `MULTNUM` use with `MULTREGT` or `MULTREGP`, will automatically copy all matrix cells to the corresponding fracture cells, so dummy fracture values could be input. For other `MULTNUM` uses, there is no copying of data from the matrix to fracture cells.

The regions for inter-region transmissibility multipliers can be output to the Print file by using the mnemonic `MULTNUM` in the `RPTGRID` keyword.

**Example**

To define 9 separate regions in a 4*4*3 field (with the maximum number of `MULTNUM` regions set equal to 9):

```
MULTNUM
1 1 2 2
1 1 2 2
3 3 3 3
4 4 4 4
5 5 5 5
6 7 8 8
6 7 8 8
6 7 8 8
16*9 /
```
MULTOUT  Indicates that output files are multiple

This indicates that output files which may be either multiple or unified are to be multiple. This is the default; unified files may be requested with the UNIFOUT keyword.

This keyword has no associated data.

<table>
<thead>
<tr>
<th>Keywords</th>
</tr>
</thead>
<tbody>
<tr>
<td>ECLIPSE 100</td>
</tr>
<tr>
<td>ECLIPSE 300</td>
</tr>
<tr>
<td>SPECIAL</td>
</tr>
<tr>
<td>RUNSPEC</td>
</tr>
<tr>
<td>GRID</td>
</tr>
<tr>
<td>EDIT</td>
</tr>
<tr>
<td>PROPS</td>
</tr>
<tr>
<td>REGIONS</td>
</tr>
<tr>
<td>SOLUTION</td>
</tr>
<tr>
<td>SUMMARY</td>
</tr>
<tr>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>
MULTOUTS  Indicates that summary output files are multiple

This indicates that summary output files which may be either multiple or unified are to be multiple. This is the default.

This keyword overwrites the behavior of the UNIFOUT keyword.

This keyword has no associated data.
**Pore volume multipliers**

The keyword should be followed by one non-negative real number for every grid block in the current input BOX. The values specified act as multipliers on the pore volumes. When used in the GRID section, these are the pore volumes calculated by ECLIPSE for each grid block. When used in the EDIT section, with ECLIPSE 300 they must act on values specified by PORV in the edit section; with ECLIPSE 100 it is recommended that they do so. When used in the SCHEDULE section, they act on the latest values produced by either GRID or EDIT. The data must be terminated by a slash (/).

Any MULTPV values that are not specified when the end of the GRID section is reached, default to 1.0.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

### Use in the EDIT section

If MULTPV is entered in the EDIT section, it acts as an additional multiplier on the pore volumes after any edited pore volumes have been read from the EDIT section. Use of MULTPV in the EDIT section in ECLIPSE 300 is only to provide modifiers to values of PORV entered explicitly in the EDIT section, and this is also the recommended practice in ECLIPSE 100. MULTPV should otherwise normally be specified in the GRID section rather than in the EDIT section. In ECLIPSE 300, ensure that the boxes for input of PORV and MULTPV match.

### Use in the SCHEDULE section

MULTPV can be used in the SCHEDULE section to modify the pore volumes during the run. Material is still conserved, so modifying the pore volume causes a step change in the pressure. Large sudden changes in the pore volume should be avoided.

This is a specialist facility, and is not recommended for general use. Note that the effect of MULTPV in the SCHEDULE section is cumulative: each time MULTPV is encountered in the SCHEDULE section, it multiplies the current pore volumes. So for a given cell, the base pore volume is multiplied by the product of all multipliers specified for that cell in the SCHEDULE section up to the current simulation time.

In ECLIPSE 100, the modified pore volume can be output to the Print file by using the mnemonic MULT in the RPTSCHED keyword.

### Example

```
---------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
   5   6   3   8   4   6   /
MULTPV
36*1.25 /
```
**MULTR**

**Transmissibility multipliers in the radial direction**

The keyword is used in radial geometry and is analogous to **MULTX** in Cartesian geometry.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the transmissibilities calculated by the program for the +R faces of each grid block.

Thus, a value of **MULTR** specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I+1, J, K).

Any **MULTR** values which are not specified when the end of the **GRID** section is reached, default to 1.0.

Grid blocks are ordered with the R axis index cycling fastest, followed by the Theta and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

In ECLIPSE 100 **MULTR** can also be used in the **EDIT** section.

---

**Note:** For ECLIPSE 100 compatibility when using ECLIPSE 300, the **SCHEDULE** section keyword **MULTTR** has been aliased to **MULTR**. However, **MULTTR** can still be used in the **SCHEDULE** section.

See also keyword **MULTR-**.

---

**Example**

```plaintext
-------- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX      5   5   3   8   4   6   /
MULTR    18*0   /
```
Transmissibility multipliers in the negative radial direction

The keyword is used in radial geometry and is analogous to MULTX-- in Cartesian geometry.

In order to use MULTR--, the negative direction transmissibility multipliers must be enabled using the keyword GRIDOPTS in the RUNSPEC section.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the transmissibilities calculated by the program for the -R face of each grid block.

Thus, a value of MULTR-- specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I-1, J, K).

Any MULTR-- values which are not specified when the end of the GRID section is reached default to 1.0.

In ECLIPSE 100, MULTR-- can also be used in the EDIT section.

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

See also the MULTR keyword.

If both MULTR and MULTR-- are specified, then the product is used. For example, if MULTR is specified for cell (I, J, K) and MULTR-- is specified for cell (I+1, J, K), then the product of the nominated MULTR and MULTR-- values is used to multiply the transmissibility between blocks (I, J, K) and (I+1, J, K).

Example

```
---- ---- ---- ---- ---- ----
    IR1-IR2 JT1-JT2 KZ1-KZ2

   BOX  5   5   3   8   4   6   /

 MULTR-- 18*0 /

 END BOX
```
**MULTREAL  Activates multi-realization licensing**

This keyword is used to activate the multiple-realization (MR) licensing. It is generated by a trusted program which creates an associated key file. The keyword cannot be added or modified by the user as this will cause the failure of validation against the key file. In such a case, the simulator will either stop or revert to the ordinary licensing depending on the value of the second item. Further information about the MR licensing can be found in "Multiple realization" in the *ECLIPSE Technical Description*.

The MULTREAL keyword is optional and should be followed by up to 2 data items, terminated by a slash (/).

1. A session identifier shared by all datasets generated simultaneously from the same case. The length of this string must not exceed 64 characters.
2. Specify if the simulator should revert to the standard licensing if the a multiple realization license is not available or this case cannot be validated successfully. The value is either **YES** or **NO**.

**DEFAULT:** YES

**Example**

```
-- Example of keyword generated by a trusted program
MULTREAL
    'C3ECCB5D-5D9A-4544-BCA4-42B13866A8B5' NO /
```
MULTREGD  Multiplies the diffusivity between flux or MULTNUM regions

The MULTREGD keyword can be used to set a diffusivity multiplier between regions. The regions must previously have been defined using keyword FLUXNUM, MULTNUM or OPERNUM in the GRID section.

If MULTNUM is to be used, then storage for the MULTNUM array must be requested by setting the second item greater than zero in the RUNSPEC section keyword GRIDOPTS. MULTREGD will apply to diffusivities between MULTNUM regions.

If FLUXNUM is to be used, then storage for the flux region data should be reserved by setting the fourth item in the RUNSPEC section keyword REGDIMS. MULTREGD will then apply to diffusivities between flux regions defined using FLUXNUM.

If OPERNUM is to be used, then storage for the OPERNUM array should be reserved by setting the seventh item in the RUNSPEC section keyword REGDIMS. MULTREGD will then apply to diffusivities between regions defined using OPERNUM.

The keyword can only be used when the Molecular Diffusion option is active (keyword DIFFUSE in the RUNSPEC section).

The MULTREGD keyword is followed by any number of data records. Each record contains the following six items of data, and is terminated with a slash (/).

1. From region number (I)
   A non-positive or defaulted value acts as a wildcard on all regions.
   DEFAULT: Negative
2. To region number (J)
   A non-positive or defaulted value acts as a wildcard on all regions.
   DEFAULT: Negative
3. Diffusivity multiplier for all diffusivities connecting regions I and J
4. Direction(s) in which to apply the multiplier:
   Either X, Y, Z, XY, YZ, XZ or XYZ.
   DEFAULT: XYZ
5. Flag governing application of multiplier to non-neighbor connections:
   NNC  Apply the diffusivity multiplier only to non-neighbor connections between regions I and J
   NONNC Do not apply the diffusivity multiplier to any non-neighbor connections between regions I and J
   ALL  Apply the diffusivity multiplier to all connections between regions I and J
   NOAQUNNC Do not apply the diffusivity multiplier to numerical aquifer connections.
   DEFAULT: ALL
6. Choice of region: MULTNUM (M), FLUXNUM (F) or OPERNUM (O)

DEFAULT: M

The set of records must end with a blank record containing only a slash (/).

Notes

The multipliers specified in the MULTREGH keyword are applied after accounting for any data entered in either the GRID or EDIT sections.

- If either region number I or J is set non-positive or defaulted, then that region identifier is taken as a wildcard and the multiplier is applied between all regions I and J, with $I \neq J$, matching the template.

- If the region numbers I and J are equal and positive, then the rock thermal conductivities within that region in the specified direction(s) are multiplied by the given value, as well as any thermal conductivities connecting any other regions.

- The keyword does not affect the non-neighbor connections between matrix and fracture blocks in a dual porosity simulation.

- Non-neighbor connections arising from numerical aquifer connections to the grid are modified by MULTREGH unless the NOAQUUNC flag is set in item 5.

- If a combination of region numbers I and J is specified twice, then the latest multiplier is applied for the rock thermal conductivities between the two regions; the multipliers are not cumulative. This is true even if different region sets are specified - it is not possible to specify a multiplier between regions I and J using MULTNUM and a different multiplier between regions I and J using FLUXNUM. However, an individual rock thermal conductivity may be modified by more than one multiplier, if it is between different region number combinations for different region sets.

- This facility can be used to modify connections between PVT regions by assigning FLUXNUM or MULTNUM in the GRID section equal to the PVT regions, and then copying FLUXNUM or MULTNUM to PVTNUM in the REGIONS section.

- This keyword cannot be used in conjunction with nested LGRs.

CAUTION: For backward compatibility reasons, while the default region set is MULTNUM, if some lines of data have a different region set specified, the first region specified is used as the default for all lines which have no region set specified. Defaulting some region sets while specifying others is not recommended.

Example

<table>
<thead>
<tr>
<th>MULTREGD</th>
<th>'From' region</th>
<th>'To' region</th>
<th>Diffusivity</th>
<th>Directions</th>
<th>NNCs</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1</td>
<td>2</td>
<td>0.5</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1</td>
<td>3</td>
<td>0.0</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>4</td>
<td>0.4</td>
<td>'XY'</td>
<td>/</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>1*</td>
<td>0.75</td>
<td>Between region 5 and any region.</td>
<td>/</td>
</tr>
</tbody>
</table>

/
MULTREGH Multiplies the rock thermal conductivity between flux or MULTNUM regions

The MULTREGH keyword can be used to set a rock thermal conductivity multiplier between regions, for the Temperature option (see "Temperature option" in the ECLIPSE Technical Description). The regions must previously have been defined using keyword FLUXNUM, MULTNUM or OPERNUM in the GRID section.

If MULTNUM is to be used, then storage for the MULTNUM array must be requested by setting the second item greater than zero in the RUNSPEC section keyword GRIDOPTS. MULTREGH applies to rock thermal conductivities between MULTNUM regions.

If FLUXNUM is to be used, then storage for the flux region data should be reserved by setting the fourth item in the RUNSPEC section keyword REGDIMS. The MULTREGH will then apply to rock thermal conductivities between flux regions defined using FLUXNUM.

If OPERNUM is to be used, then storage for the OPERNUM array should be reserved by setting the seventh item in the RUNSPEC section keyword REGDIMS. MULTREGT will then apply to rock thermal conductivities between regions defined using OPERNUM.

This keyword can only be used when the Temperature option is active (keyword TEMP in the RUNSPEC section), and the rock thermal conductivities must have been defined using keyword THCONR in the GRID section.

The MULTREGH keyword is followed by any number of data records. Each record contains the following six items of data, and is terminated with a slash (/).

1. From region number (I)
   A non-positive or defaulted value acts as a wildcard on all regions.
   DEFAULT: Negative
2. To region number (J)
   A non-positive or defaulted value acts as a wildcard on all regions.
   DEFAULT: Negative
3. Rock thermal conductivity multiplier for all thermal conductivities connecting regions I and J
4. Direction(s) in which to apply the multiplier:
   Either X, Y, Z, XY, YZ, XZ or XYZ.
   DEFAULT: XYZ
5. Flag governing application of multiplier to non-neighbor connections:
   NNC  Apply the rock thermal conductivity multiplier only to non-neighbor connections between regions I and J
   NONNC Do not apply the rock thermal conductivity multiplier to any non-neighbor connections between regions I and J
   ALL  Apply the thermal conductivity multiplier to all connections between regions I and J
   NOAQUNNC Do not apply the rock thermal conductivity multiplier to numerical aquifer connections.
6. Choice of region: MULTNUM (M), FLUXNUM (F) or OPERNUM (O)

DEFAULT: M

The set of records must end with a blank record containing only a slash (/).

Notes

The multipliers specified in the MULTREGH keyword are applied after accounting for any data entered in either the GRID or EDIT sections.

- If either region number I or J is set non-positive or defaulted, then that region identifier is taken as a wildcard and the multiplier is applied between all regions I and J, with $I \neq J$, matching the template.

- If the region numbers I and J are equal and positive, then the rock thermal conductivities within that region in the specified direction(s) are multiplied by the given value, as well as any thermal conductivities connecting any other regions.

- The keyword does not affect the non-neighbor connections between matrix and fracture blocks in a dual porosity simulation.

- Non-neighbor connections arising from numerical aquifer connections to the grid are modified by MULTREGH unless the NOAQNUNC flag is set in item 5.

- If a combination of region numbers I and J is specified twice, then the latest multiplier is applied for the rock thermal conductivities between the two regions; the multipliers are not cumulative. This is true even if different region sets are specified - it is not possible to specify a multiplier between regions I and J using MULTNUM and a different multiplier between regions I and J using FLUXNUM. However, an individual rock thermal conductivity may be modified by more than one multiplier, if it is between different region number combinations for different region sets.

- This facility can be used to modify connections between PVT regions by assigning FLUXNUM or MULTNUM in the GRID section equal to the PVT regions, and then copying FLUXNUM or MULTNUM to PVNUM in the REGIONS section.

- This keyword cannot be used in conjunction with nested LGRs.

CAUTION: For backward compatibility reasons, while the default region set is MULTNUM, if some lines of data have a different region set specified, the first region specified is used as the default for all lines which have no region set specified. Defaulting some region sets while specifying others is not recommended.

Example

<table>
<thead>
<tr>
<th>MULTREGH</th>
<th>'From' region</th>
<th>'To' region</th>
<th>Conductivity multiplier</th>
<th>Directions</th>
<th>NNCs</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1</td>
<td>2</td>
<td>0.5</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1</td>
<td>3</td>
<td>0.0</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>4</td>
<td>0.4</td>
<td>'XY'</td>
<td>/</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>1*</td>
<td>0.75</td>
<td>/</td>
<td>Between region 5 and any region.</td>
</tr>
</tbody>
</table>

Keywords

1378
MULTREGP  Multiplies the pore volumes for specific flux or MULTNUM region

The MULTREGP keyword can be used to set a pore volume multiplier for a specific region. The regions must previously have been defined using keyword FLUXNUM, MULTNUM or OPERNUM in the GRID section.

If MULTNUM is to be used, then storage for the MULTNUM array must be requested by setting the 2nd item greater than zero in the RUNSPEC section keyword GRIDOPTS. MULTREGP applies to pore volumes to the MULTNUM regions.

If FLUXNUM is to be used, then storage for the flux region data should be reserved by setting the fourth item in the RUNSPEC section keyword REGDIMS. MULTREGP will then apply to pore volumes between flux regions defined using FLUXNUM.

If OPERNUM is to be used, then storage for the OPERNUM array should be reserved by setting item 7 in the RUNSPEC section keyword REGDIMS. MULTREGP will then apply to pore volumes between flux regions defined using OPERNUM.

The MULTREGP keyword is followed by any number of data records. Each record contains the following three items of data, and is terminated with a slash (/).

1. Region number (I)  
   DEFAULT: Negative
2. Pore volume multiplier for region I
3. Choice of region: MULTNUM (M), FLUXNUM (F) or OPERNUM (O)  
   DEFAULT: M

The set of records must end with a blank record containing only a slash (/).

Notes

• The multipliers specified in the MULTREGP keyword are applied after accounting for any MULTPV values specified in the GRID or EDIT sections.

• If a region number is specified twice, then the latest multiplier is applied for that region; the multipliers are not cumulative. This is true even if different region sets are specified - it is not possible to specify a multiplier for region 1 using MULTNUM and a different multiplier for region 1 using FLUXNUM. However, an individual grid block’s pore volume may be modified by more than one multiplier, if it is in different region numbers for different region sets.

• The multipliers for specific regions can be output to the Print file using the mnemonic MULTREGP in the RPTGRID keyword.

• For MULTREGP to be used in the EDIT section, PORV values should also be specified in the EDIT section.

• If the pore volume values are not specified, a warning message will be produced.

• Only positive region numbers will be able to modify the pore volume given the pore volume multiplier assigned to that region. A pore volume multiplier assigned to a zero or negative region number will have no effect on the pore volumes.
• This keyword cannot be used in conjunction with nested LGRs.

**CAUTION:** For backward compatibility reasons, while the default region set is MULTNUM, if some lines of data have a different region set specified, the first region specified is used as the default for all lines which have no region set specified. Defaulting some region sets while specifying others is not recommended.

**Example**

```
MULTREGP
  -- Region  Pore Volume Multiplier  Region Set
  1        0.5                    M/
  2        0.0                    F/
  4        0.4                    M/
  5        0.75                   O/
/```

Keywords
MULTREGT Multiplies the transmissibility between flux or MULTNUM regions

The MULTREGT keyword can be used to set a transmissibility multiplier between regions. The regions must previously have been defined using keyword FLUXNUM, MULTNUM or OPERNUM in the GRID section.

If MULTNUM is to be used, then storage for the MULTNUM array must be requested by setting the second item greater than zero in the RUNSPEC section keyword GRIDOPTS. MULTREGT applies to transmissibilities between MULTNUM regions.

If FLUXNUM is to be used, then storage for the flux region data should be reserved by setting the fourth item in the RUNSPEC section keyword REGDIMS. MULTREGT will then apply to transmissibilities between flux regions defined using FLUXNUM.

If OPERNUM is to be used, then storage for the OPERNUM array should be reserved by setting the item 7 in the RUNSPEC section keyword REGDIMS. MULTREGT will then apply to transmissibilities between regions defined using OPERNUM.

The MULTREGT keyword is followed by any number of data records. Each record contains the following six items of data, and is terminated with a slash (/).

1. From region number (I)
   A non-positive or defaulted value acts as a wildcard on all regions.
   DEFAULT: Negative
2. To region number (J)
   A non-positive or defaulted value acts as a wildcard on all regions.
   DEFAULT: Negative
3. Transmissibility multiplier for all transmissibilities connecting regions I and J
4. Direction(s) in which to apply the multiplier:
   Either X, Y, Z, XY, YZ, XZ or XYZ.
   DEFAULT: XYZ
5. Flag governing application of multiplier to non-neighbor connections:
   NNC        Apply the transmissibility multiplier only to non-neighbor connections between regions I and J
   NONNC      Do not apply the transmissibility multiplier to any non-neighbor connections between regions I and J
   ALL        Apply the transmissibility multiplier to all connections between regions I and J
   NOAQUNNC   Do not apply the transmissibility multiplier to numerical aquifer connections.
   DEFAULT: ALL
6. Choice of region: MULTNUM (M), FLUXNUM (F) or OPERNUM (O)
   DEFAULT: M
The set of records must end with a blank record containing only a slash (/).

When using MULTREGT in the SCHEDULE section with ECLIPSE 100, only the first three items (and item 6 if required) should be specified; the remaining items assume that values are specified in the GRID section or are defaulted. The effect of using MULTREGT in the SCHEDULE section is cumulative. Hence if MULTREGT is specified twice for the same connection between regions, the product of the transmissibility multipliers is used.

The effect of MULTREGT in the GRID section is not cumulative.

Notes

• The multipliers specified in the MULTREGT keyword are applied after accounting for any values of MULTX, MULTY, MULTZ, or MULTFLT entered in either the GRID or EDIT sections.

• Similarly, the multipliers specified in this keyword are applied after accounting for any values of MULTX-, MULTY- or MULTZ-.

• If either region number I or J is set non-positive or defaulted, then that region identifier is taken as a wildcard and the multiplier is applied between all regions I and J, with I ≠ J, matching the template.

• If the region numbers I and J are equal and positive, then the transmissibilities within that region in the specified direction(s) are multiplied by the given value, as well as any transmissibilities connecting any other regions.

• The keyword does not affect the non-neighbor connections between matrix and fracture blocks in a dual porosity simulation.

• Non-neighbor connections arising from numerical aquifer connections to the grid are modified by MULTREGT unless the NOAQNUNNC flag is set in item 5.

• In ECLIPSE 100, if a combination of region numbers I and J is specified twice, then the latest multiplier is applied for the transmissibilities between the two regions; the multipliers are not cumulative. This is true even if different region sets are specified - it is not possible to specify a multiplier between regions I and J using MULTNUM and a different multiplier between regions I and J using FLUXNUM. However, an individual transmissibility may be modified by more than one multiplier, if it is between different region number combinations for different region sets.

• In ECLIPSE 100, this facility can be used to modify connections between PVT regions by assigning FLUXNUM or MULTNUM in the GRID section equal to the PVT regions, and then copying FLUXNUM or MULTNUM to PVTNUM in the REGIONS section.

• In order to use MULTREGT in the EDIT section, TRANX, TRANY, TRANZ values should also be specified in the EDIT section. If the transmissibility values are not specified, a warning message will be produced.

• The multipliers between regions can be output to the print file using the mnemonic MULTREGT in the RPTGRID keyword.

• This keyword cannot be used in conjunction with nested LGRs.

• In ECLIPSE 300, when specifying region numbers for individual LGR cells for use with MULTREGT, global host cell values which are greater than 1 will overwrite local cell region numbers which are equal to 1, even if these have been explicitly set rather than defaulted. Region numbers greater than 1 should be used for LGR cells which have host cells not in region 1.
**CAUTION:** For backward compatibility reasons, while the default region set is MULTNUM, if some lines of data have a different region set specified, the first region specified is used as the default for all lines which have no region set specified. Defaulting some region sets while specifying others is not recommended.

### Example

```plaintext
MULTREGT
  -- 'From' region   'To' region   Transmissibility   Directions   NNCs
  multiplier

1    2    0.5   /   
1    3    0.0   /   
4    4    0.4   'XY'   /   
5    1*   0.75 / Between region 5 and any region.
```

**Keywords**

1383
**MULTSAVE**  Indicates that save files are multiple

This indicates that ECLIPSE 300 SAVE files, which may be either multiple or unified, are to be multiple. This is the default; unified SAVE files may be requested with the UNIFSAVE keyword.

This keyword also allows the writing of SAVE files to be controlled. The keyword contains an integer data item.

1. Select how multiple SAVE files are used:
   - -1 Suppress SAVE files.
   - 0 Overwrite the first SAVE file at each step.
   - 1 Produce SAVE files at each SAVE keyword.

   DEFAULT: 0

The setting of MULTSAVE affects the operation of the AUTOSAVE keyword.

This keyword is not available in ECLIPSE 100, and will be ignored.

ECLIPSE 300 does not allow a SAVE file to be written during a parallel run. Fast restarts are not supported for parallel runs in ECLIPSE 300.
MULTSIG

Dual porosity matrix-fracture coupling multiplier

If the dual porosity option is being used (keyword DUALPORO in the RUNSPEC section), and the matrix-fracture coupling transmissibilities have been specified in the grid section through the SIGMA or SIGMAV keywords, then this keyword can be used to specify a multiplier in the SCHEDULE section which will modify further these matrix-fracture coupling transmissibilities.

If this keyword is used, a single multiplier is applied to the entire current grid. To specify values for individual cells, use the MULTSIGV keyword.

If this keyword is used multiple times or it is used in combination with MULTSIGV then the effect of the multipliers will be cumulative.

The keyword should be followed by one single positive real number, which will be applied to every cell in the current grid.

UNITs: dimensionless

Example

```
MULTSIG
  0.5 /
```
MULTSIGV  

**Dual porosity matrix-fracture coupling multiplier**

If the dual porosity option is being used (keyword `DUALPORO` in the `RUNSPEC` section), and the matrix-fracture coupling transmissibilities have been specified in the grid section through the `SIGMA` or `SIGMAV` keywords, then this keyword can be used to specify a multiplier in the `SCHEDULE` section which will modify further these matrix-fracture coupling transmissibilities.

The keyword should be followed by one positive real number for every grid block in the current box within the top `NDIVIZ/2` layers of the grid (that is, the matrix cells). Any values input in the bottom half of the grid (the lower `NDIVIZ/2` layers) will be ignored.

To specify a single value for all cells, use the `MULTSIG` keyword.

If this keyword is used multiple times or it is used in combination with `MULTSIG` then the effect of the multipliers will be cumulative.

**UNITS:** dimensionless

---

**Example**

```
BOX 1 8 1 1 1 2 /
MULTSIGV 0.5 0.5 0.5 0.5 0.5 0.5 0.3 0.3 0.3 0.3 0.3 0.3 /
```
Transmissibility multipliers in the theta direction

The keyword is used in radial geometry and is analogous to MULTY in Cartesian geometry.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the transmissibilities calculated by the program for the +Theta faces of each grid block.

Thus, a value of MULTTHT specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I, J+1, K).

Any MULTTHT values that are not specified when the end of the GRID section is reached, default to 1.0.

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

In ECLIPSE 100, MULTTHT can also be used in the EDIT section.

Note: For ECLIPSE 100 compatibility when using ECLIPSE 300, the SCHEDULE section keyword MULTTTHT has been aliased to MULTTHT. However, MULTTTHT can still be used in the SCHEDULE section as documented in previous releases.

See also keyword MULTTHT-.

Example

--------------- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX          4  6  3  3  4  6  /
MULTTHT     9*0.01 /
ENDBOX

Keywords

1387
**MULTTHT-**

**Transmissibility multipliers in the negative theta direction**

The keyword is used in radial geometry and is analogous to **MULTY-** in Cartesian geometry.

In order to use **MULTTHT-**, the negative direction transmissibility multipliers must be enabled using the keyword **GRIDOPTS** in the **RUNSPEC** section.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the transmissibilities calculated by the program for the -theta faces of each grid block.

Thus, a value of **MULTTHT-** specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I, J-1, K).

Any **MULTTHT-** values that are not specified when the end of the **GRID** section is reached default to 1.0.

In ECLIPSE 100, **MULTTHT-** can also be used in the **EDIT** section.

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

See also the **MULTTHT** keyword.

If both **MULTTHT** and **MULTTHT-** are specified, then the product is used. For example, if **MULTTHT** is specified for cell (I, J, K) and **MULTTHT-** is specified for cell (I, J+1, K), then the product of the nominated **MULTTHT** and **MULTTHT-** values is used to multiply the transmissibility between blocks (I, J, K) and (I, J +1, K).

### Example

```
-------- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX      4   6   3   3   4   6   /
MULTTHT  9*0.01 /
ENDBOX
```
Transmissibility multipliers in X-direction

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the transmissibilities calculated by the program for the +X face of each grid block or specified directly using \texttt{TRANX} in the \texttt{EDIT} section.

Thus, a value of \texttt{MULTX} specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I+1, J, K).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

See also keyword \texttt{MULTX-}.

Non-neighbor connections

Any non-neighbor connections generated due to faults have transmissibilities that reflect the \texttt{MULTX} values. Non-neighbor connections entered explicitly using the \texttt{NNC} keyword are not affected.

Use in the GRID section

If \texttt{MULTX} is entered in the \texttt{GRID} section, then it applies only to the transmissibilities calculated by the program using the information in the \texttt{GRID} section, and not to any transmissibilities entered directly in the \texttt{EDIT} section using the \texttt{TRANX} keyword. Any \texttt{MULTX} values that are not specified when the end of the \texttt{GRID} section is reached, default to 1.0.

Use in the EDIT section (ECLIPSE 100 only)

If \texttt{MULTX} is entered in the \texttt{EDIT} section, then it acts as an additional multiplier on the transmissibilities after the reading of any modified transmissibilities from the \texttt{EDIT} section. Any non-neighbor connections are modified to reflect the edited values of \texttt{MULTX}.

Use of the \texttt{MULTX} multiplier in the \texttt{EDIT} section is not recommended in general, except as a modifier to values of \texttt{TRANX} entered explicitly in the \texttt{EDIT} section. The keyword \texttt{MULTX} should normally be specified in the \texttt{GRID} section rather than in the \texttt{EDIT} section.

Use in the SCHEDULE section

\texttt{MULTX} can be used in the \texttt{SCHEDULE} section to modify the transmissibilities during the run. Well connection transmissibility factors are not affected by this keyword (instead, use \texttt{WPIMULT} for example, for this purpose).

Note that the effect of \texttt{MULTX} in the \texttt{SCHEDULE} section is cumulative. In other words, each time \texttt{MULTX} is encountered in the \texttt{SCHEDULE} section, it multiplies the current X-direction transmissibility. So for a given cell, the base X-direction transmissibility is multiplied by the product of all multipliers specified for that cell in the \texttt{SCHEDULE} section up to the current simulation time.

The modified transmissibilities can be output to the Print file by using the mnemonic \texttt{MULT} in the \texttt{RPTSCHED} keyword.

In ECLIPSE 300, if values are defaulted or set negative, the transmissibilities for the corresponding cells are not modified.
**Note:** For ECLIPSE 100 compatibility when using ECLIPSE 300, the `SCHEDULE` section keyword `MULTTX` has been aliased to `MULTX`. However, `MULTTX` can still be used in the `SCHEDULE` section.

### Example

This example multiplies the transmissibility values in a box by 0.2

```plaintext
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX      5  5  3  8  4  6  /
MULTX 18*0.2 /
ENDBOX
```
**MULTX-**  

**Transmissibility multipliers in the negative X-direction**

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the transmissibilities calculated by the program for the -X face of each grid block or specified directly using TRANX in the EDIT section.

Thus, a value of MULTX- specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I-1, J, K).

In order to use MULTX-, the negative direction transmissibility multipliers must be enabled using the keyword GRIDOPTS in the RUNSPEC section.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

See also the MULTX keyword.

If both the MULTX keyword and MULTX- are specified, then the product is used. For example, if MULTX is specified for the block (I, J, K) and MULTX- is specified for the block (I+1, J, K), then the program uses the product of the nominated MULTX and MULTX- values to multiply the calculated transmissibility between the two adjacent blocks.

**Non-neighbor connections**

Any non-neighbor connections generated due to faults have transmissibilities that reflect the MULTX and MULTX- values. A fault non-neighbor connection between a cell (I, J, K1) and a cell (I+1, J, K2) in an adjacent column has a transmissibility factor of MULTX for block (I, J, K1) or MULTX- for block (I+1, J, K2) or the product of the two. Non-neighbor connections entered explicitly using the NNC keyword are not affected.

**Use in the GRID section**

If MULTX- is entered in the GRID section, then it applies only to the transmissibilities calculated by the program using the information in the GRID section, and not to any transmissibilities entered directly in the EDIT section using the TRANX keyword. Any MULTX- values that are not specified when the end of the GRID section is reached, default to 1.0.

**Use in the EDIT section (ECLIPSE 100 only)**

If MULTX- is entered in the EDIT section, then it acts as an additional multiplier on the transmissibilities after reading any modified transmissibilities from the EDIT section. Any non-neighbor connections are modified to reflect the edited values of MULTX-.

Use of the MULTX- multiplier in the EDIT section is not recommended in general, except as modifiers to values of TRANX entered explicitly in the EDIT section. The keyword MULTX- should normally be specified in the GRID section rather than in the EDIT section.
Use in the SCHEDULE section

MULTX- can be used in the SCHEDULE section to modify the transmissibilities during the run. Well connection transmissibility factors are not affected by this keyword (instead, use WPIMULT for example, for this purpose).

The modified transmissibilities can be output to the print file by using the mnemonic MULT in the RPTSCHED keyword.

In ECLIPSE 300, if values are defaulted or set negative, the transmissibilities for the corresponding cells are not modified.

Example

```
------------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
    5 5 3 8 4 6 /
MULTX-
   18*0 /
ENDBOX
```
MULTY

Transmissibility multipliers in Y-direction

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the transmissibilities calculated by the program for the +Y face of each grid block or specified directly using TRANY in the EDIT section.

Thus, a value of MULTy specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I, J+1, K).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

See also keyword MULTy-.

Non-neighbor connections

Non-neighbor connections entered explicitly using the NNC keyword are not affected.

Use in the GRID section

If MULTy is entered in the GRID section, then it applies only to the transmissibilities calculated by the program using the information in the GRID section, and not to any transmissibilities entered directly in the EDIT section using the TRANY keyword. Any MULTy values which are not specified when the end of the GRID section is reached, default to 1.0.

Use in the EDIT section (ECLIPSE 100 only)

If MULTy is entered in the EDIT section, then it acts as an additional multiplier on the transmissibilities after the reading of any modified transmissibilities from the EDIT section. Any non-neighbor connections will be modified to reflect the edited values of MULTy.

Use of the MULTy multiplier in the EDIT section is not recommended in general, except as a modifier to values of TRANZ entered explicitly in the EDIT section. The keyword MULTy should normally be specified in the GRID section rather than in the EDIT section.

Use in the SCHEDULE section

MULTy can be used in the SCHEDULE section to modify the transmissibilities during the run. Well connection transmissibility factors are not affected by this keyword (instead, use WPIMULT for example, for this purpose).

Note that the effect of MULTy in the SCHEDULE section is cumulative. In other words, each time MULTy is encountered in the SCHEDULE section, it multiplies the current Y-direction transmissibility. So for a given cell, the base Y-direction transmissibility is multiplied by the product of all multipliers specified for that cell in the SCHEDULE section up to the current simulation time.

The modified transmissibilities can be output to the Print file by using the mnemonic MULt in the RPTSCHEd keyword.

In ECLIPSE 300, if values are defaulted or set negative, the transmissibilities for the corresponding cells are not modified.
**Note:** For ECLIPSE 100 compatibility when running ECLIPSE 300, the **SCHEDULE** section keyword **MULTTX** has been aliased to **MULTY**. However, **MULTTY** can still be used in the **SCHEDULE** section.

**Example**

This example multiplies the transmissibility values in a box by zero.

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX      5  5   3  8   4   6 /
MULTY    18*0.0 /
ENDBOX
```
MULTY-Transmissibility multipliers in the negative Y-direction

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the transmissibilities calculated by the program for the -Y face of each grid block or specified directly using TRANY in the EDIT section.

Thus, a value of MULTY– specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I, J-1, K).

In order to use MULTY–, the negative direction transmissibility multipliers must be enabled using the keyword GRIDOPTS in the RUNSPEC section.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

See also the MULTY keyword.

If both the MULTY keyword and MULTY– are specified, then the product is used. For example, if MULTY is specified for the block (I, J, K) and MULTY– is specified for the block (I, J-1, K), then the program uses the product of the nominated MULTY and MULTY– values to multiply the calculated transmissibility between the two adjacent blocks.

Non-neighbor connections

Any non-neighbor connections generated due to faults have transmissibilities that reflect the MULTY and MULTY– values. A fault non-neighbor connection between a cell (I, J, K1) and a cell (I, J+1, K2) in an adjacent column has a transmissibility factor of MULTY for block (I, J, K1) or MULTY– for block (I, J +1, K2) or the product of the two. Non-neighbor connections entered explicitly using the NNC keyword are not affected.

Use in the GRID section

If MULTY– is entered in the GRID section, then it applies only to the transmissibilities calculated by the program using the information in the GRID section, and not to any transmissibilities entered directly in the EDIT section using the TRANY keyword. Any MULTY– values that are not specified when the end of the GRID section is reached, default to 1.0.

Use in the EDIT section (ECLIPSE 100 only)

If MULTY– is entered in the EDIT section, then it acts as an additional multiplier on the transmissibilities after reading any modified transmissibilities from the EDIT section. Any non-neighbor connections are modified to reflect the edited values of MULTY–.

Use of the MULTY– multiplier in the EDIT section is not recommended in general, except as modifiers to values of TRANY entered explicitly in the EDIT section. The keyword MULTY– should normally be specified in the GRID section rather than in the EDIT section.

Keywords
Use in the SCHEDULE section

MULTY- can be used in the SCHEDULE section to modify the transmissibilities during the run. Well connection transmissibility factors are not affected by this keyword (instead, use WPIMULT for example, for this purpose).

The modified transmissibilities can be output to the print file by using the mnemonic MULT in the RPTSCHED keyword.

In ECLIPSE 300, if values are defaulted or set negative, the transmissibilities for the corresponding cells are not modified.

Example

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX 5 5 3 8 4 6 /
MULTY- 18*0 /
ENDBOX
```
**MULTZ**

**Transmissibility multipliers in Z-direction**

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the transmissibilities calculated by the program for the +Z face of each grid block or specified directly using `TRANZ` in the `EDIT` section.

Thus, a value of `MULTZ` specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I, J, K+1).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

See also keyword `MULTZ-`.

**Non-neighbor connections**

Non-neighbor connections entered explicitly using the `NNC` keyword are not affected.

**Use in the GRID section**

If `MULTZ` is entered in the GRID section, then it applies only to the transmissibilities calculated by the program using the information in the GRID section, and **not** to any transmissibilities entered directly in the EDIT section using the `TRANZ` keyword. Any `MULTZ` values which are not specified when the end of the GRID section is reached, default to 1.0.

**Use in the EDIT section (ECLIPSE 100 only)**

If `MULTZ` is entered in the EDIT section, then it acts as an additional multiplier on the transmissibilities after the reading of any modified transmissibilities from the EDIT section. Any non-neighbor connections will be modified to reflect the edited values of `MULTZ`.

Use of the `MULTZ` multiplier in the EDIT section is **not** recommended in general, except as a modifier to values of `TRANZ` entered explicitly in the EDIT section. The keyword `MULTZ` should normally be specified in the GRID section rather than in the EDIT section.

**Use in the SCHEDULE section**

`MULTZ` can be used in the SCHEDULE section to modify the transmissibilities during the run. Well connection transmissibility factors are **not** affected by this keyword (instead, use `WPIMULT` for example, for this purpose).

Note that the effect of `MULTZ` in the SCHEDULE section is cumulative. In other words, each time `MULTZ` is encountered in the SCHEDULE section, it multiplies the current Z-direction transmissibility. So for a given cell, the base Z-direction transmissibility is multiplied by the product of all multipliers specified for that cell in the SCHEDULE section up to the current simulation time.

The modified transmissibilities can be output to the Print file by using the mnemonic `MULT` in the `RPTSCHED` keyword.

In ECLIPSE 300, if values are defaulted or set negative, the transmissibilities for the corresponding cells are not modified.
Note: For ECLIPSE 100 compatibility when running ECLIPSE 300, the SCHEDULE section keyword MULTTX has been aliased to MULTZ. However, MULTTZ can still be used in the SCHEDULE section as documented in previous releases.

Example

This example multiplies the transmissibility values in a box by 1000.

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
   1 11 1 19 2 2 /
MULTZ
   209*1000.0 /
ENDBOX
```
MULTZ-

**Transmissibility multipliers in the negative Z-direction**

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified act as multipliers on the transmissibilities calculated by the program for the -Z face of each grid block or specified directly using `TRANZ` in the `EDIT` section.

Thus, a value of `MULTZ-` specified for block (I, J, K) multiplies the transmissibility between blocks (I, J, K) and (I, J, K-1).

In order to use `MULTZ-`, the negative direction transmissibility multipliers must be enabled using the keyword `GRIDOPTS` in the `RUNSPEC` section.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

See also the `MULTZ` keyword.

If both the `MULTZ` keyword and `MULTZ-` are specified, then the product is used. For example, if `MULTZ` is specified for the block (I, J, K) and `MULTZ-` is specified for the block (I, J, K+1), then the program uses the product of the nominated `MULTZ` and `MULTZ-` values to multiply the calculated transmissibility between the two adjacent blocks.

**Non-neighbor connections**

Any non-neighbor connections generated due to pinchouts have transmissibilities that reflect the `MULTZ` values for the upper of the two cells connected across a pinchout, and the `MULTZ-` values for the lower of the two cells. A pinchout non-neighbor connection in column (I, J) between layers K1 and K2, with K2 > K1, has its calculated transmissibility multiplied by the value of `MULTZ` for cell (I, J, K1) and `MULTZ-` for cell (I, J, K2). Non-neighbor connections entered explicitly using the `NNC` keyword are not affected.

**Use in the GRID section**

If `MULTZ-` is entered in the `GRID` section, then it applies only to the transmissibilities calculated by the program using the information in the `GRID` section, and not to any transmissibilities entered directly in the `EDIT` section using the `TRANZ` keyword. Any `MULTZ-` values that are not specified when the end of the `GRID` section is reached, default to 1.0.

**Use in the EDIT section (ECLIPSE 100 only)**

If `MULTZ-` is entered in the `EDIT` section, then it acts as an additional multiplier on the transmissibilities after reading any modified transmissibilities from the `EDIT` section. Any non-neighbor connections are modified to reflect the edited values of `MULTZ-`.

Use of the `MULTZ-` multiplier in the `EDIT` section is not recommended in general, except as modifiers to values of `TRANZ` entered explicitly in the `EDIT` section. The keyword `MULTZ-` should normally be specified in the `GRID` section rather than in the `EDIT` section.

---

**Keywords**

```
<table>
<thead>
<tr>
<th>ECLIPSE 100</th>
<th>ECLIPSE 300</th>
</tr>
</thead>
<tbody>
<tr>
<td>SPECIAL</td>
<td></td>
</tr>
<tr>
<td>RUNSPEC</td>
<td></td>
</tr>
<tr>
<td>GRID</td>
<td></td>
</tr>
<tr>
<td>EDIT</td>
<td>PROPS</td>
</tr>
<tr>
<td>REGIONS</td>
<td>SOLUTION</td>
</tr>
<tr>
<td>SUMMARY</td>
<td></td>
</tr>
<tr>
<td>SCHEDULE</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>
```
Use in the SCHEDULE section

MULTZ- can be used in the SCHEDULE section to modify the transmissibilities during the run. Well connection transmissibility factors are not affected by this keyword (instead, use WPIMULT for example, for this purpose).

The modified transmissibilities can be output to the Print file by using the mnemonic MULT in the RPTSCHED keyword.

In ECLIPSE 300, if values are defaulted or set negative, the transmissibilities for the corresponding cells are not modified.

Example

```plaintext
---------- IX1-IX2 JY1-JY2 KZ1-KZ2
   BOX   1 11  1 19  2 2 /
   MULTZ-
       209*1000 /
   ENDBOX
```
**MW**

**Molecular weights**

In a run with \( N_c \) components, this keyword associates a mean molecular weight with each component. The keyword should be followed by \( N_c \) values.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword **TABDIMS**. The use of multiple EoS regions is discussed in “Equations of state regions” in the *ECLIPSE Technical Description*.

**UNITS**: kg/kg-M (**METRIC**), lb/lb-M (**FIELD**), gm/gm-M (**LAB**), kg/kg-M (**PVT-M**)

**DEFAULT**: There are no component-wise default values. The users are recommended not to default these to the internal default value.

**Example**

Molecular weights for a four-component study:

```
MW
18 32 45.6 78.0 /
```
MWDETAIL  Molecular weights for detailed components

In a run with \( N_d \) detailed components, this keyword associates a mean molecular weight with each of the detailed components. Lumped simulations are enabled with the LUMPDIMS keyword in the RUNSPEC section; further details can be found in "Delumping and lumping" in the ECLIPSE Technical Description.

The keyword should be followed by a record of \( N_d \) values, terminated with a slash, for each reservoir equation of state region. The values represent the molecular weight of each detailed component, in the order in which the detailed components are defined in the LUMPING keyword.

If multiple equations of state are being used, a record of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

UNITs: kg/kg-M (METRIC), lb/lb-M (FIELD), gm/gm-M (LAB), kg/kg-M (PVT-M)

Example

Molecular weights for a study with 8 detailed components and one EoS region:

```
MWDETAIL
18 32 45.6 78.0 56.2 37.6 67.8 81.0 /
```
Molecular weights for surface EoS

This optional keyword supplies molecular weights for use in the surface calculations. The default, if this is not supplied, is for these to follow the reservoir values supplied with MW.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

UNITS: kg/kg-M (METRIC), lb/lb-M (FIELD), gm/gm-M (LAB), kg/kg-M (PVT-M)
Molecular weights of water components

In a run with $N_w$ water components, this keyword associates a mean molecular weight with each component.

The keyword should be followed by $N_w$ values.

UNITs: kg/kg-M (METRIC), lb/lb-M (FIELD), gm/gm-M (LAB), kg/kg-M (PVT-M)

DEFAULT: 18.0

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword `TABDIMS`. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

See "Water components" in the ECLIPSE Technical Description for further details of the water component option.

Example

Molecular weights for a two-component study:

```
MWW
18.0 18.5 /
```
Molecular weights for water components for surface EoS

This optional keyword supplies molecular weights of water components for use in the surface calculations. The default, if this is not supplied, is for these to follow the reservoir values supplied with MWW.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

UNITS: kg/kg-M (METRIC), lb/lb-M (FIELD), gm/gm-M (LAB), kg/kg-M (PVT-M)
N

This section contains an alphabetic listing of keywords starting with the letter N.
# NARROW

## Narrow columns for Run Summary output

This keyword increases the number of columns per page in the Run Summary output (generated by using the `RUNSUM` keyword).

This keyword can be used to output ECLIPSE 100 data in the pre-97A format, or ECLIPSE 300 data in the pre-2000A format.

The NARROW keyword has no associated data.

### Example

```
NARROW
```

### Keywords

- ECLIPSE 100
- ECLIPSE 300
- SPECIAL
- RUNSPEC
- GRID
- EDIT
- PROPS
- REGIONS
- SOLUTION
- SUMMARY
- SCHEDULE
NCOMPS  Confirm number of components

This keyword may be read in the PROPS section to confirm the number of components in a compositional run. NCOMPS contains an item of data which specifies the number of components to be used.

This keyword is not usually required, as the number of components will have been specified by keyword COMPS in the RUNSPEC section. If present it is read, and an error message is produced if the item’s value does not match that specified with COMPS. The purpose of NCOMPS is to ensure that included files produced by PVT have the correct number of components, and PVO files automatically contain this keyword.

Example

Confirm that twelve components are present:

```
  NCOMPS
  12  /
```
NCONSUMP  Gas consumption for extended network nodes

This keyword is used to specify a constant gas consumption rate at a particular node in the extended network (see “The extended network model” in the ECLIPSE Technical Description), and optionally to assign the consumption also to a group in the GRUPTREE hierarchy. It may be used as an alternative to the GCONSUMP keyword when the extended network model is employed (keyword NETWORK present in the RUNSPEC section).

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Node name
2. Gas consumption rate at the node
   UNITS: sm³/day (METRIC), Mcf/day (FIELD), scc/hr (LAB), sm³/day (PVT-M)
   DEFAULT: Zero
3. Name of the group from which the node's consumption should also be removed
   DEFAULT: Not removed from any group

**Note:**

- If item 3 is defaulted, the node's gas consumption is **not** removed from a group, even if the node is a source node and thus corresponds to a group. The default action is therefore different from that in GCONSUMP.
- The group defined in item 3 replaces any group previously defined for the same purpose by this keyword or item 8 in keyword NETCOMPA.

**CAUTION:** Do not use both NCONSUMP and GCONSUMP to declare consumption at corresponding nodes and groups; this could double the consumption.

See also keyword GCONSUMP.

**Example**

```
NCONSUMP
  PLAT-A  20     PLAT-A  /
  PLAT-B  15     PLAT-B  /
/
```
NEFAC

Efficiency factors for extended network nodes

The keyword may be used to specify an efficiency factor at a node in the extended network (see "The extended network model" in the ECLIPSE Technical Description). This may be useful if a group efficiency factor has been applied with keyword GEFAC to a group which does not correspond to a network node; NEFAC may then be used to apply the efficiency factor to an appropriate node in the network, if desired. Alternatively it may be used to set a node's efficiency factor (which governs its contribution to downstream nodes in the network) to a value different from the efficiency factor that has been set for its corresponding group with keyword GEFAC. See "Well and group efficiency factors" in the ECLIPSE Technical Description for further discussion on the handling of efficiency factors.

**CAUTION:** Care should be taken if you are also using the GEFAC keyword with item 3 set to YES, if the node name and the group name are the same. If GEFAC is entered after NEFAC then the nodal efficiency factors gets reset by GEFAC. On the other hand, using NEFAC does not change a group's efficiency factor, even if it has the same name as the node. The efficiency values assigned to the groups/nodes can be checked by using the WELSPECS mnemonic in the RPTSCHED keyword.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. **Node name**

2. **Efficiency factor for the node**
   
   This is the fractional time for which the node is operational. For example, if the node is 'down' for 10 percent of the time, its efficiency factor will be 0.9.

   The efficiency factor must be greater than zero.

   DEFAULT: 1.0

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

The efficiency factors for nodes not specified with this keyword are defaulted to 1.0.

**Example**

```
NEFAC
   NODE4  0.9 /
/
```

---

**Keywords**

1410
NEI

Specify composition for non-equilibrium initialization

This keyword may be used to generate non-equilibrium initialization composition tables internally. It is followed by a sample hydrocarbon composition for each equilibration region number. This is used to generate consistent oil and gas compositions for each cell, by flashing to the cell pressure - effectively a constant composition expansion.

For the THERMAL option, the simulator tries to match the oil, water and gas saturations in each equilibration region. If the specified compositions cannot match the phase saturations, then the components that have the highest gas phase mole fraction have their compositions adjusted. See "Initialization with NEI" in the ECLIPSE Technical Description.

ECLIPSE Thermal. For the Mixed Hydrostatic Equilibration option, initial saturations should be specified together with items 1 and 2 of the EQUIL keyword and the NEI keyword. For more information, refer to "Initializing the study" in the ECLIPSE Technical Description.

Example

```
NEI
0.0247 0.2516 0.2407 0.0733 0.4097 /
```
NETBALAN  Network balancing calculation instructions

In ECLIPSE 100 this keyword forces a network balancing calculation to be performed at the next timestep, and controls the frequency of subsequent balancing calculations in the network option (see "Network option" in the ECLIPSE Technical Description). The keyword can also reset the convergence tolerance and iteration limit for the network balancing calculations, and the calculation of the THP for subsea completion manifold groups under production rate control. In addition, the user may enter instructions to control the end of timestep balancing error by limiting the timestep size.

In ECLIPSE 300 this keyword just controls the convergence tolerance and iteration limit. The network will automatically be rebalanced during the first NUPCOL Newton iterations of each timestep (equivalent to a negative entry in item 1), and there is no facility to control the timestep size according to the end of timestep balancing error.

The keyword should be followed by a line containing the following data items, ending with a slash (/). If the record is terminated early, the remaining items assume their default values. Default values can be specified before the slash by null value indicators (n*, where n is the number of consecutive items to be defaulted). Defaulted items retain their previous values; initial default values are shown below.

1. Network balancing interval (ECLIPSE 100 only)
   > 0.0  The network is balanced at the beginning of each timestep that starts after the specified interval has elapsed since the previous balancing calculation.Balancing errors build up if the reservoir conditions change significantly over this balancing interval.

   = 0.0  The network is balanced at the beginning (that is the zeroth Newton iteration) of every timestep. Balancing errors are only significant if the reservoir conditions change appreciably over the timestep.

   < 0.0  The network will be balanced in each of the first NUPCOL Newton iterations of every timestep. (The value of NUPCOL may be set using keyword NUPCOL or keyword GCONTOL.) The magnitude of the negative value is not significant. This option may cause the timestep to require more Newton iterations to converge, as the wells’ THP limits are changed at each balancing calculation. However, the balancing errors at the end of the timestep will generally be small, and are less than the value set in item 2 if the timestep converges within NUPCOL Newton iterations.

   UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

   DEFAULT: 0.0

2. Convergence tolerance for network nodal pressures

   In ECLIPSE 300 the convergence tolerance for the network is a factor of 10 smaller than the tolerance set here for the network to reservoir coupling. In ECLIPSE 100 these iterations are performed simultaneously and have the same convergence tolerance.

   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

   ECLIPSE 100 DEFAULT: 0.1 bars (METRIC), 1.45 psi (FIELD), 0.09869 atm (LAB, PVT-M).

   ECLIPSE 300 DEFAULT: 1E-5 bars (METRIC), 1.45E-4 psi (FIELD), 9.869 E-6 atm (LAB, PVT-M).

3. Maximum number of iterations allowed in the network balancing calculation

   In ECLIPSE 300 this item governs both the iterations for the network solution and the iterations for coupling the network to the reservoir.
4. Convergence tolerance for calculating the THP for subsea completion manifold groups under rate control, and for calculating the pressure drop across automatic chokes controlling a group’s production rate (ECLIPSE 100 only).

The quantity required is the fraction of each group’s rate target that is an acceptable error.

The group’s production rate may differ from the target by more than this tolerance if the timestep takes more than NUPCOL Newton iterations to converge - see keyword NUPCOL or GCONTOL.

If the value entered here is greater than the tolerance fraction entered in item 1 of keyword GCONTOL, the value entered in GCONTOL is used.

DEFAULT: 0.01 Which converges the iterations when the group’s rate target is met within one percent.

5. Maximum number of iterations allowed in the calculation of the THP for manifold groups under rate control (ECLIPSE 100 only).

DEFAULT: 10

Note: The maximum number of iterations for automatic chokes is governed by item 3, since their pressure drops are calculated during the network balancing iterations.

6. Target value for the largest branch balancing error at the end of each timestep (ECLIPSE 100 only).

The balancing error is defined as the difference between the pressure drop along the branch when the network was last balanced, and the pressure drop calculated using the current flow rates at the end of the timestep. The largest error in any branch is printed at the end of each timestep. If a value is entered for the target error, ECLIPSE attempts to adjust the timestep length so as to keep the balancing error at approximately its target value, subject of course to any other constraints on the timestep. Timesteps limited by the target balancing error can be identified by the mnemonic NETW printed in the timestep information line.

The target error should be large enough to allow for the convergence tolerances in items 2 and 4.

UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

DEFAULT: Infinity (1.0E20) which leaves the timestep sizes unaffected.

7. Maximum permitted value for the largest branch balancing error at the end of the timestep (ECLIPSE 100 only).

If the maximum permitted balancing error is exceeded, ECLIPSE chops the timestep and rebalance the network. Since timestep chops can severely increase the running time of a simulation, a maximum permitted balancing error should only be used with caution, and its value should be several times larger than the target error in item 6.

UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

DEFAULT: Infinity (1.0E20)

8. Minimum size for timesteps limited by network balancing error (ECLIPSE 100 only)

A large balancing error is sometimes unavoidable during the timestep in which a well dies. If a target or maximum balancing error has been set in items 6 and 7, it is advisable to set a suitable minimum timestep size to prevent a large reduction in step size when wells die. (The minimum step size does not affect timesteps limited by other reasons, for example time truncation error or convergence difficulty.)
UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

DEFAULT: TSMINZ in keyword TUNING.

If the keyword is omitted, all items take their default values.

The keyword may be used any number of times in the SCHEDULE section. Defaulted items retain their previous values.

**Examples - ECLIPSE 100**

**Example 1**

Sets the target and maximum balancing error, with a minimum step size override of 10 days. The network is balanced at the beginning of every timestep, by default.

```
NETBALAN 5 10.0 50.0 10.0 /
```

**Example 2**

Cause the network to be balanced in each of the first NUPCOL Newton iterations of every timestep:

```
NETBALAN -1.0 /
```
NETCOMPA  Defines automatic compressors in extended network

The keyword defines automatic compressors or pumps in the production network when the extended network model is employed (keyword NETWORK entered in the RUNSPEC section). Each compressor/pump occupies a branch of the network, and its operation is modeled by changing the VFP table number or ALQ value of the branch.

See "Automatic compressors" in the ECLIPSE Technical Description for further details.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Inlet node name of the branch where the compressor is located.
2. Outlet node name of the branch where the compressor is located.
3. Name of group whose production rate this compressor will respond to.
   The compressor will be activated whenever this group fails to meet its target production rate set for the phase in item 4. If the group does not have a target set, but has a guide rate defined in GCONPROD and is under FLD control from a higher level group, then the compressor is activated when the group cannot make its share of the higher level group's production target.
   If other actions to increase the group's production rate may also be performed (for example opening a new well from the drilling queue), the order in which the possible actions are performed can be defined with the PRORDER keyword.
   DEFAULT: '' implies the compressor is permanently off
4. Phase whose production rate this compressor responds to.
   OIL or GAS
   DEFAULT: GAS
5. VFP table number for the branch when the compressor is on.
   When the compressor is switched on this replaces the table number previously specified in keyword BRANPROP, which is applied whenever the compressor is off.
   Zero will leave the original VFP table number unchanged.
   DEFAULT: 0
6. Artificial Lift Quantity to be applied when the compressor is fully on.
   When the compressor is fully on this replaces the ALQ value previously specified in keyword BRANPROP, which is applied whenever the compressor is off. Multilevel compressors operating at an intermediate level has an intermediate value of the ALQ (see items 9, 10 and 11).
   The artificial lift quantity may be regarded as the compressor or pump power, according to the definition used when the table was calculated (for example with VFP i).
   UNITS: These depend on what the ALQ was meant to represent when the table was created. Refer to section "Record 6:ALQ values" in the description of keyword VFPDROD.
   DEFAULT: 0.0
7. Rate of gas consumption by the compressor when fully on.

When the compressor is fully on this is extracted from the inlet node (item 1) to account for the gas required to power the compressor. Multilevel compressors operating at an intermediate level have an intermediate value of gas consumption (see items 9, 10 and 11). The consumption rate is applied in addition to any consumption specified for the node with keyword NCONSUMP or GCONSUMP.

UNITS: $\text{sm}^3/\text{day (METRIC)}$, Mscf/day (FIELD), scc/hr (LAB), $\text{sm}^3/\text{day (PVT-M)}$

DEFAULT: 0.0

Note: If you wish the consumed gas to be removed from the outlet node of the branch, reverse the order of the inlet and outlet nodes in items 1 and 2.

8. Name of the group from which the compressor's gas consumption (item 7) should also be extracted.

Gas consumption from the group hierarchy is taken into account for group reinjection control and sales gas control, and also by the Gas Field Operations Model. (See keyword GCONSUMP.)

DEFAULT: ' ' the gas is only extracted from the network, and not the group hierarchy

Note: The group defined here replaces any group previously defined for the same purpose using this keyword or item 3 of keyword NCONSUMP.

9. Compressor type.

PERM Will remain on permanently after being turned on.

Keyword COMPOFF has no effect.

TEMP Can be turned off manually at any time with the COMPOFF keyword (for example whenever the target production rate is reduced). It is turned on automatically again as soon as it is needed.

If the gas field operations model is in use, the compressor is turned off automatically whenever the contract group's target rate decreases. It is turned on automatically again as soon as it is needed.

MULT As TEMP, but with multiple compression levels.

Compression is increased one level at a time until the group's target rate is met.

10. The number of compression levels in a multilevel compressor

(an integer > 0)

This item must be set only if item 9 is set to MULT. Otherwise it must be defaulted.

Any number of levels may be requested, but having a large number of levels slows the run down as each level must be tried in turn.

If there are $N$ levels, the ALQ and consumption rate at level $i$ are:

$$ALQ_i = ALQ_1 + (ALQ_{N-1} - ALQ_1) \left(\frac{i-1}{N-1}\right)$$

Eq. 3.122

$$CONS_i = CONS_N \cdot \frac{i}{N}$$

Eq. 3.123
where $ALQ_N$ is defined in item 6, $CONSN_N$ is defined in item 7 and $ALQ_1$ is defined in item 11.

The VFP table number will be the value set in item 5 when the compressor is operating at its first level and above.

11. Artificial lift quantity at level 1 of multi-level compressor, $ALQ_1$.

This item must be set only if item 9 is set to 'MULT'. Otherwise it must be defaulted.

UNITs: These depend on what the ALQ was meant to represent when the table was created. Refer to section "Record 6:ALQ values" in the description of keyword VFPPRD.

12. Compressor switching sequence number.

This prescribes the order in which compressors are turned on if there are two or more compressors responding to the same group (item 3). When a group fails to meet its rate target, the compressor with the lowest sequence number that responds to the group is activated. If the group still cannot meet its target when this is fully on, but has other compressors that respond to it, these other compressors are turned on in increasing order of their sequence number.

If two compressors anywhere in the network share the same sequence number, they are both turned on at the same time, even if they respond to different groups. In this way it is possible to turn on all compressors simultaneously whenever compression is needed anywhere in the field. All multi-level compressors with the same sequence number have their levels incremented simultaneously.

DEFAULT: a value 1 larger than the previous maximum sequence number.

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also keywords PRORDER and COMPOFF.

**Note:** An automatic compressor cannot be present if the branch's ALQ is made equivalent to the oil or gas surface density in item 5 of keyword BRANPROP.

**Example**

The following example defines two compressors responding to the gas production rate of groups 'PL-A' and 'PL-B' respectively. The second one has three compression levels, and consumes gas from the network and from group 'G-B3' in the GRUPTREE hierarchy.

```
NETCOMPA
  -- Inlet Outlet Target Phase VFP ALQ Cons Cons Type Num ALQ Seq
  -- node node group tab rate group rate group
  N4 N3 PL-A GAS 1* 50.0 1* 1* TEMP                  /
  N6 N5 PL-B GAS 1* 100.0 1000 G-B3 MULT 3 40.0      /
```

**Keywords**
**NETWORK**

Sets dimensions for extended network model

This keyword is required if the extended network model is used (keywords NODEPROP and BRANPROP; see "The extended network model" in the ECLIPSE Technical Description). It turns on the extended network model facilities and reserves storage space for the network description.

The data must be terminated by a slash (/).

1. **NODMAX** The maximum number of nodes in the network model.
2. **NBRMAX** The maximum number of branches in the network model.
3. **NBCMAX** (ECLIPSE 300 only) The maximum number of branches connected to any one node in the network model.
   
   **DEFAULT:** 20

In ECLIPSE 100, the NETWORK keyword should **not** be entered if the standard network model is to be used for the production network (keyword GRUPNET).

In ECLIPSE 300, the NETWORK keyword should **not** be entered if the standard network model is to be used (keywords GRUPNET and GNETINJE). The standard and extended network models cannot both be used in the same run.

**Example**

```
NETWORK
  9 8 /
```
**NEWTON**

**Outputs the iteration count to the summary file**

This keyword requests that the number of non-linear iterations used for each timestep should be output to the Summary file.

The **NEWTON** keyword has no associated data.

**Example**

```
NEWTON
```

**Keywords**

<table>
<thead>
<tr>
<th>x</th>
<th>ECLIPSE 100</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>ECLIPSE 300</td>
</tr>
<tr>
<td>x</td>
<td>SPECIAL</td>
</tr>
<tr>
<td>x</td>
<td>RUNSPEC</td>
</tr>
<tr>
<td>x</td>
<td>GRID</td>
</tr>
<tr>
<td>x</td>
<td>EDIT</td>
</tr>
<tr>
<td>x</td>
<td>PROPS</td>
</tr>
<tr>
<td>x</td>
<td>REGIONS</td>
</tr>
<tr>
<td>x</td>
<td>SOLUTION</td>
</tr>
<tr>
<td>x</td>
<td>SUMMARY</td>
</tr>
<tr>
<td>x</td>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>
NEWTRAN  Specifies block corner transmissibilities

The use of NEWTRAN implies that transmissibilities are calculated from cell corner positions. This permits the automatic calculation of fault transmissibilities, but it may not be back-compatible with existing simulators using block-center calculations.

NEWTRAN has no associated data.

This is the default method for grids specified using the corner point method (COORD and ZCORN keywords). If the grid is specified using DX (or DR), DY (or DTHETA) and DZ then by default the transmissibilities are calculated using the block-center quantities unless NEWTRAN is specified. (See also keywords OLDTRAN and OLDTRANR.)

See "Transmissibility calculations" in the ECLIPSE Technical Description.

Note: When NEWTRAN is specified in a block centered grid, the grid is first converted to a 'fake' corner point representation composed of cuboidal cells with the dimensions specified by DX (or DR), DY (or DTHETA) and DZ before calculating transmissibilities. For a dipping block-centered grid, this creates unrealistic fault non-neighbor connections that cannot exist in the true block-centered grid. Thus NEWTRAN should not normally be specified in a block-centered grid.

Keywords

<table>
<thead>
<tr>
<th>x</th>
<th>ECLIPSE 100</th>
<th>ECLIPSE 300</th>
<th>SPECIAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>RUNSPEC</td>
<td></td>
<td></td>
</tr>
<tr>
<td>x</td>
<td>GRID</td>
<td>EDIT</td>
<td>PROPS</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td>REGIONS</td>
<td>SOLUTION</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>SUMMARY</td>
</tr>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>

1420
Sets a maximum value for the next timestep

This is an alias for the keyword NEXTSTEP.
Sets a maximum value for the next timestep

The keyword is used to declare a maximum possible value for the next timestep. This provides a simpler alternative to using the TUNING keyword to set this value.

The keyword should be followed by the required items of data, terminated by a slash(/):

1. The maximum length of the next timestep.
   
   UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT–M)

2. Apply this value at every subsequent report step? YES or NO.
   
   DEFAULT: NO
   
   The keyword does not restrict the length of the subsequent timesteps following the next timestep if this item is set to NO.

Typically, this keyword may be used to force a short timestep immediately following a large well rate change, to reduce the danger of a timestep chop (see also TMAXWC in Record 1 of the TUNING keyword).

Example

This example

```
NEXTSTEP
1 /
```

replaces:

```
TUNING
1 /
/
```
**NEXTSTPL**  

**Sets a maximum value for the next local timestep**

The keyword is used to declare a maximum possible value for the next local timestep in each of the local grid systems. This provides a simpler alternative to using the `TUNINGL` keyword to set this value.

The keyword should be followed by the required items of data, terminated by a slash (/):

1. **The maximum length of the next local timestep.**
   
   The value is applied to all local grid systems.
   
   **UNITS:** day (`METRIC`), day (`FIELD`), hr (LAB), day (PVT–M)

2. **Apply this value at every subsequent report step? YES or NO.**
   
   **DEFAULT:** NO
   
   The keyword does not restrict the length of the subsequent timesteps following the next timestep if this item is set to NO.

Typically, this keyword may be used to force a short timestep immediately following a large well rate change, to reduce the danger of a timestep chop.

**Example**

This example

```
NEXTSTPL
  1 /
```

replaces

```
TUNINGL
  1 /
  / /
```

NGASREM  Removes gas from a node in the extended network

This keyword can be used in the ECLIPSE 300 extended network (constructed with keywords BRANPROP and NODEPROP, with keyword NETWORK present in the RUNSPEC section) to remove gas from the network at a specified node. The amount of gas to be removed can be expressed as either a constant rate or a fraction of the rate flowing through that node. The removal of gas affects the pressure loss in the downstream network branches as their VFP tables are looked up with a smaller gas fraction. However, this is the only effect that it does have. It does not affect the reported group production rates or the operation of any group gas rate targets/limits, reinjection or group economic limits. A possible use for NGASREM is to remove gas from a network node to account for its removal from the production stream by the GRUPFUEL keyword.

This keyword is not available in ECLIPSE 100, where GCONSUMP or NCONSUMP should be used instead.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Node name
2. Maximum rate of gas removal from the node
   The specified rate of gas will be removed, subject to a maximum fraction of the node's gas flow entered in item 3.
   UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)
   DEFAULT: Infinity
3. Maximum fraction of the node's gas flow to be removed
   The specified fraction of the node's gas flow is removed, subject to a maximum rate entered in item 2.
   DEFAULT: 1.0

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

**Note:** The defaults in items 2 and 3 only apply to nodes that have been named in item 1 of this keyword. All other nodes retain their initial values of 0.0 for both quantities, so that no gas is removed from them.

**CAUTION:** Since gas removal only affects the pressure losses in the downstream branches, an error is reported if the node in item 1 has no outflow branch.

**Examples**

**Example 1**

Remove 20000 units/day of gas from the node. Item 3 defaults to 1.0 to ensure that the gas removal rate does not exceed the flow rate of gas through the node.
Example 2

Remove one tenth of the gas that flows through the node. Item 2 defaults to infinity so that there is no other limit on the removal rate.

```
NGASREM
  NODE-5  20000  
  
ECLIPSE Reference Manual
Keywords

1425`
NINENUM

Identifies the extent of the nine-point region

The NINENUM keyword indicates regions in which to apply the nine-point discretization scheme for the NINEPOIN option (activated in the GRID section). This keyword is entirely optional when using the nine-point scheme; if it is not specified then the nine-point discretization is applied everywhere.

The NINENUM keyword should be followed by one integer for every grid block in the current input box specifying the discretization flag for each grid block. The flag should be set either to 1 to indicate that the nine-point discretization is to be employed, or to zero in regions where the standard finite-difference discretization is required (five points in 2D, seven points in 3D).

Any grid blocks not explicitly assigned a NINENUM value is defaulted to a value of 1. You must set the NINENUM value explicitly to zero for any regions where the nine-point scheme is not required.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

Note: This keyword is not compatible with the local grid refinement option.

Example

To apply the nine-point scheme only in a 2*2*2 region in a 4*4*3 field:

```
NINENUM
1 1 0 0
1 1 0 0
0 0 0 0
1 1 0 0
1 1 0 0
0 0 0 0
0 0 0 0
16*0 /
```
Select the nine-point option

This keyword selects a nine-point transmissibility option, which adds diagonal transmissibility values to the grid as non-neighbor connections in the XY plane. The method of obtaining these extra transmissibility values is described in "Nine-point schemes" in the ECLIPSE Technical Description. The total x- and y-transmissibilities over the grid are preserved, but the addition of the diagonal terms may limit grid orientation effects when fluids are injected at adverse mobility ratios.

The keyword has no associated data.
NINEXZ

Select the nine-point option in the XZ plane

This keyword selects a nine-point transmissibility option which adds diagonal transmissibility values to the grid as non-neighbor connections in the XZ plane. The method of obtaining these extra transmissibility values is described in "Nine-point schemes" in the ECLIPSE Technical Description. The total x- and z-transmissibilities over the grid are preserved, but the addition of the diagonal terms may limit grid orientation effects when fluids are injected at adverse mobility ratios.

The keyword has no associated data.
NINEYZ  Select the nine-point option in the YZ plane

This keyword selects a nine-point transmissibility option which adds diagonal transmissibility values to the grid as non-neighbor connections in the YZ plane. The method of obtaining these extra transmissibility values is described in "Nine-point schemes" in the ECLIPSE Technical Description. The total y- and z-transmissibilities over the grid are preserved, but the addition of the diagonal terms may limit grid orientation effects when fluids are injected at adverse mobility ratios.

The keyword has no associated data.
### Options for the discretized matrix (E100) or multi porosity (E300) models

For ECLIPSE 100 the keyword is used to control options associated with the discretized matrix dual porosity model. The model must be activated by including the \texttt{NMATRIX} keyword in the \texttt{RUNSPEC} section.

For ECLIPSE 300 the keyword means that the model (see "Multi porosity" in the \textit{ECLIPSE Technical Description}), activated with \texttt{NMATRIX}, is assigned grid data automatically (similar to the discretized matrix Dual Porosity Model of ECLIPSE 100). This means that grid data only needs to be supplied as for a standard dual porosity case.

The \texttt{NMATOPTS} keyword is optional and should be followed by some or all of the following data items, terminated by a slash (/).

1. The assumed geometry of the matrix sub-cells.
   
   Either \texttt{LINEAR}, \texttt{RADIAL} or \texttt{SPHERICAL}
   
   In ECLIPSE 300 there is an additional possibility - \texttt{UNIFORM}. This produces a uniform grid within the matrix cells.
   
   A further alternative possibility is \texttt{VERTICAL}, which is used to model gravity drainage, whereas the previous options model transient effects in the flow from matrix to fracture. \texttt{VERTICAL} uses the matrix cells as a vertical discretization of an individual block of matrix and connects them all to the fracture. See "Vertical discrete matrix gravity drainage model" in the \textit{ECLIPSE Technical Description}. If this option is used then the keyword \texttt{GRAVDRB} must have been specified in the \texttt{RUNSPEC} section.
   
   In ECLIPSE 100, for any discretized matrix option, the minimum number of global cells in the \texttt{X} direction (keyword \texttt{DIMENS}) is 2.
   
   In all cases except \texttt{VERTICAL}, the matrix sub-cells only communicate with the fracture system through the outer sub-cell. However, the pressure (and saturation) response from the matrix depends on the assumed geometry of the system. In the case of \texttt{VERTICAL} geometry, all matrix sub-cells also connect directly with the fracture, and the physical response from the matrix depends on all connections. In this case, the geometry is uniform as described in the Technical Description.
   
   \texttt{DEFAULT}: \texttt{LINEAR}

2. The fraction of the fracture pore volume (or matrix block volume with ECLIPSE 300 - see item 3) is to be used for the size of the outermost matrix sub-cell. It is not used if \texttt{UNIFORM} or \texttt{VERTICAL} is selected. \texttt{VERTICAL} assumes uniform distribution of the matrix volume in the same way as \texttt{UNIFORM}.
   
   Once the size of the outermost sub-cell and the number of sub-cells in each matrix cell have been defined, ECLIPSE automatically generates a logarithmic grid within the matrix cells. The maximum and minimum growth factors over the entire model are reported as a message.
   
   To model accurately the initial pressure transient response of the matrix cells, the outermost matrix sub-cell needs to be reasonably small compared to the fracture system.
   
   \texttt{DEFAULT}: 0.1

3. Method for controlling the partitioning (ECLIPSE 300 only).
   
   Either \texttt{FPORV} or \texttt{MBLKV}
FPORV  the fractional value in item 2 is used to construct the size of the first partitioned block from the fracture pore volume.

MBLKV  the fractional value in item 2 is used to construct the size of the first partitioned block from the matrix block volume.

Using the time-dependent sorption model in ECLIPSE 300 with the COAL option, the first pore volume of the matrix sub-grid might be different from the given fractional value since only some of the sub-grid cells might have a pore volume. When there is no pore volume in the matrix (only coal volume) the MBLKV method will be used instead.

DEFAULT: FPORV

For further information see "Discretized matrix model (ECLIPSE 100)" in the ECLIPSE Technical Description.

Example

NMATOPTS
RADIAL  0.05 /
**NMATRIX**

**Use the discretized matrix or multi-porosity model**

Defines the number of matrix pore systems.

In ECLIPSE 100 this keyword activates the discretized matrix dual porosity model, in which the matrix is subdivided into smaller cells to allow the modeling of transient effects within the matrix cells (without **NMATOPTS**, or with the first 3 (ECLIPSE 100) or 4 (ECLIPSE 300) **NMATOPTS** options), or to create a more detailed representation of gravity drainage (**NMATOPTS** keyword, **VERTICAL** option). See the **NMATOPTS** keyword in the **GRID** section and "Dual porosity" in the *ECLIPSE Technical Description* for more details.

In ECLIPSE 300 this keyword activates the multi-porosity option. See "Multi porosity" in the *ECLIPSE Technical Description*.

**Note:** In ECLIPSE 100, explicit initialization of the internal sub-matrix cells is not possible, and the **EQUIL** keyword must be used with **NMATRIX** so that the sub-matrix cells are properly initialized.

With this keyword in ECLIPSE 100, the minimum number of global cells in the X direction (keyword **DIMENS**) is 2.

With this keyword in ECLIPSE 100, the additional Z-planes for the matrix sub-cells are stored internally and are not part of the deck as described in the **DIMENS** or **CARFIN** keywords. For this simulator, these keywords should only describe the dual porosity model size.

By contrast, with this keyword in ECLIPSE 300, the additional Z-planes for the matrix sub-cells are explicitly represented in the **DIMENS** or **CARFIN** keywords. For this simulator, the NZ values in these keywords should be multiples of the number of porosities, that is (NMATRIX+1).

This model is not compatible with the use of block to block connections (**BTOBALFA** or **BTOBALFV**) or with the use of **DPNUM** to create a single porosity region in a dual porosity run.

This keyword should not be used for triple-porosity modeling. The **TRPLPORO** keyword should be used instead.

The keyword should be followed by one data item, defining the number of sub-cells required within each original matrix cell. The data item should be followed by a slash (/).

**Example**

| NMATRIX | 6 / |

**Keywords**

1432
Set non-neighbor connection values explicitly

Non-neighbor connections are usually generated from within the program by faults when corner point geometry is used; from the dual porosity or dual permeability options; or from numerical aquifer options.

This keyword enables a specified transmissibility to be set up between any two cells in the grid.

The keyword consists of a line of data for each required non-neighbor connection, specifying the cells to be linked and the required transmissibility value. The cell coordinates are specified in natural order.

Each line of data must be terminated by a slash (/). After the last non-neighbor connection is defined, an additional slash on a new line terminates the list. It is possible to specify this keyword more than once in the grid data.

The values in each data record are:

1. I-index of first cell joined to the non-neighbor connection (IX)
2. J-index of first cell joined to the non-neighbor connection (IY)
3. K-index of first cell joined to the non-neighbor connection (IZ)
4. I-index of second cell joined to the non-neighbor connection (JX)
5. J-index of second cell joined to the non-neighbor connection (JY)
6. K-index of second cell joined to the non-neighbor connection (JZ)
7. Transmissibility value of the non-neighbor connection (TRAN)
   
   UNITS: cP.rm³/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm³/day/atm (PVT-M)

   DEFAULT: 0.0

8. This item depends on which simulator is being used:

   **ECLIPSE 100** Saturation table number to be associated with flow from the first to the second cell (IST1)

   **ECLIPSE 300** Diffusivity value of the non-neighbor connection

   DEFAULT: 0.0

9. This item depends on which simulator is being used:

   **ECLIPSE 100** Saturation table number to be associated with flow from the second to the first cell (IST2)

   **ECLIPSE 300** Usage deprecated. Enter thermal conduction transmissibility value using item 15 of this keyword.

10. Pressure table number to be associated with flow from the first to the second cell (IPT1)

11. Pressure table number to be associated with flow from the second to the first cell (IPT2)

12. The face associated with flow from the first to the second cell (ZF1)
   
   Choose one of: X+, X-, Y+, Y-, Z+, Z-
This information is used only by the VE option (keyword VE in the RUNSPEC section).

13. The face associated with flow from the second to the first cell (ZF2)

Choose one of: X+, X-, Y+, Y-, Z+, Z-

This information is used only by the VE option (keyword VE in the RUNSPEC section).

14. Diffusivity value of the non-neighbor connection (DIFF)

This information is used by the molecular diffusion option (keyword DIFFUSE in the RUNSPEC section) and the tracer diffusion option (keyword TRDIF in the PROPS section).

UNITOS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: 0.0

If a non-default value is input for the diffusivity in ECLIPSE 300 then this value will overwrite any value input in data item 8.

15. This item depends on which simulator is being used:

ECLIPSE 300 only

Thermal conduction transmissibility value of the non-neighbor connection

Used by the THERMAL option.

UNITOS: kJ/day/K (METRIC), Btu/day/°R (FIELD), J/hr/K (LAB), kJ/day/K (PVT-M)

DEFAULT: 0.0

ECLIPSE 100 only

1/(area*porosity) (Only used in calculations involving the velocity, see DISPERSE for example).

UNITOS: m² (METRIC), ft² (FIELD), cm² (LAB), m² (PVT-M)

DEFAULT: 0.0

16. The area associated with the NNC

Used by the non-Darcy flow option (keyword VDFLOW in the PROPS section).

UNITOS: m² (METRIC), ft² (FIELD), cm² (LAB), m² (PVT-M)

DEFAULT: No non-Darcy effect.

17. The linking permeability associated with the NNC.

Used by the non-Darcy flow option (keyword VDFLOW in the PROPS section).

UNITOS: mD (METRIC), mD (FIELD), mD (LAB), mD (PVT-M)

**Notes**

ECLIPSE 100

The flow between two cells is solved using Newtonian iteration based on the permeability and geometry of the cells. For user input NNCs, we estimate the geometry, that is DXi and DXj using item 7, 16 and 17 as follows:
\[ DX_j = DX_i \]
\[ DX_i = \frac{C_{Darcy} AK_{link}}{2T} \]

where

- \( C_{Darcy} \) is a unit conversion factor
- \( A \) is the surface area given in item 16
- \( K_{link} \) is the linking permeability given in item 17
- \( T \) is the transmissibility given in item 7.

**ECLIPSE 100**

The last seven items are optional only when \( \text{VDFLOW} \) is not active. Therefore, if the slash is placed after item 7, the table numbers are defaulted to those for the cells joined. If directional relative permeabilities are employed, the positive X-direction table number is used.

**ECLIPSE 300**

Data items 8 and 9 can still be used to maintain back compatibility; it is, however, recommended to use the equivalent data items 14 and 15.

**ECLIPSE 100**

A non-neighbor connection between two cells which are actually neighbors (or between which ECLIPSE 100 would form a non-neighbor connection in the absence of the NNC keyword) is allowed: this simply adds to the existing transmissibility. In addition, any NNC input with transmissibility less than 0.000001 is ignored. Furthermore, if a transmissibility value of 1.0D-20 or less is entered, then a warning message is issued, and the NNC is ignored. Connections to inactive cells result in a warning, and are ignored.

**Examples**

**Example 1**

Two non-neighbor connections are defined, one between cells (1, 1, 1) and (3, 1, 1), and one between cells (1, 2, 1) and (4, 2, 1), each of transmissibility 0.1972.

```
 NNC
  1 1 1 3 1 1 0.1972 / 
  1 2 1 4 2 1 0.1972 / 
 / 
```

**Example 2**

General format for ECLIPSE 100

```
 NNC
 IX IY IZ JX JY JZ TRAN IST1 IST2 IPT1 IPT2 ZF1 ZF2 DIFF / 
 . . 
 / 
```

**Keywords**

1435
NNCGEN

Set non-neighbor connection values between cells in any two grids explicitly

Non-neighbor connections are usually generated from within the program by faults when corner point geometry is used; from the dual porosity or dual permeability options; or from numerical aquifer options.

This keyword enables a specified transmissibility to be set up between any two cells in any two grids.

The keyword consists of a line of data for each required non-neighbor connection, specifying the cells to be linked and the required transmissibility value. The cell coordinates are specified in natural order.

The values in each data record are:

1. Grid of first cell
   DEFAULT: GLOBAL
2. I-index of first cell joined to the non-neighbor connection (IX)
3. J-index of first cell joined to the non-neighbor connection (IY)
4. K-index of first cell joined to the non-neighbor connection (IZ)
5. Grid of second cell
   DEFAULT: GLOBAL
6. I-index of second cell joined to the non-neighbor connection (JX)
7. J-index of second cell joined to the non-neighbor connection (JY)
8. K-index of second cell joined to the non-neighbor connection (JZ)
9. Transmissibility value of the non-neighbor connection (TRAN)
   UNITS: cP.rm³/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm³/day/atm (PVT-M)
   DEFAULT: 0.0
10. Diffusivity value of the non-neighbor connection
    UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
    DEFAULT: 0.0
11. Thermal conduction transmissibility value of the non-neighbor connection, used by the Thermal option (keyword THERMAL in the RUNSPEC section)
    UNITS: kJ/day/K (METRIC), Btu/day/R (FIELD), J/hr/K (LAB), kJ/day/K (PVT-M)
    DEFAULT: 0.0

Each line of data must be terminated by a slash (/). After the last non-neighbor connection is defined, an additional slash on a new line terminates the list.

It is possible to specify this keyword more than once in the GRID section.
Example

Two non-neighbor connections are defined, one between cells (1, 1, 1) in the global grid and (3, 1, 1) in an LGR called LGR1, and one between cells (1, 2, 1) in LGR1 and (4, 2, 1) in an LGR called LGR2. Each has transmissibility 0.1972.

```
NNCGEN
'GLOBAL' 1 1 1  'LGR1' 3 1 1 0.1972 /
'LGR1' 1 2 1  'LGR2' 4 2 1 0.1972 /
/ 
```
NNEWTF

Activate the non-Newtonian fluid option

This keyword is used to activate the non-Newtonian fluid option when the polymer option is activated (see POLYMER keyword). It is followed by two integers, specifying the number of tables (NTHRBL), and the maximum number of rows in a table (NLNHBL). The integers are used in setting the FHERCHBL keyword. It must be terminated with a slash (/).

For more details please refer to the "Non-Newtonian fluid flows" option in the ECLIPSE Technical Description.

**items:**

1. NTHRBL: number of tables
   
   DEFAULT: NTPVT (number of PVT tables - see item 2 of keyword TABDIMS)

2. NLNHBL: maximum number of rows in a table
   
   DEFAULT: 2

**Example**

For NTHRBL=2 tables of NLNHBL=4 rows:

```
NNEWTF
2 4 /
```
NOAIMLGR  Suppress making all refinement cells implicit in AIM runs

In 2002A this became the default setting so the keyword has no function. LGRs can be treated fully implicitly in an AIM run by using the second item in AIMCON.
NOCASC  Use linear solver tracer algorithm

This keyword forces the linear solver algorithm to be used for solving single phase tracers. Otherwise, by default, the cascade algorithm is used.

The keyword has no associated data.

See "Tracer tracking" in the ECLIPSE Technical Description for further information.
**NOCIRCLE** Turn off completion of circle option for local radial refinements

By default, radial refinements have the circle completed automatically in their set of grid blocks. Therefore, this keyword should only be used if you do **not** wish the circle to be completed for a particular radial refinement in the theta- or Y-direction.

This keyword only has an effect if placed within a specific radial refinement. If the circle completion option is required to be turned off within a particular radial LGR, the NOCIRCLE keyword should be placed within the LGR’s RADFIN / ENDFIN bracket. Any other radial LGRs present will still have their circles completed.

The NOCIRCLE keyword is followed by one data record containing a single character string, ending with a slash (/).

The character string should be one of the following:

**YES** Turn off the circle completion within this radial refinement

**NO** Keep the circle completed within this radial refinement

**DEFAULT:** **YES**

*Note:* This keyword should not be confused with the GRID section keyword CIRCLE. The CIRCLE keyword is used to request completion of circle in a radial geometry (global) grid, not within radial refinements.

Local grid refinement is described in "Local grid refinement and coarsening" in the *ECLIPSE Technical Description*.

**Example**

```
RADFIN
LGR1 1 1 2 3 6 1 4 1* 0.2 /
NOCIRCLE
YES /
ENDFIN
```
NODEPROP  Defines the extended network node properties

The keyword must be used to define the properties of the network nodes when the extended network model is employed (keyword NETWORK in the RUNSPEC section). Keyword BRANPROP must have previously been entered. The top node of each network gathering tree (the terminal node) must be a fixed pressure node. Only the fixed pressure node, and other nodes with non-default properties need to be specified here. Nodes with default properties may be omitted from the NODEPROP keyword data.

The NODEPROP keyword is followed by any number of records, each containing the following items of data and terminated with a slash (/). The set of records must end with a blank record, containing only a slash.

1. In ECLIPSE 300, the node name
   Node names should be unique both within a network and across multiple production and injection networks.

2. Fixed pressure for the terminal node
   Default this item, or set it negative, if the node is not a terminal node in the network.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

3. Flag indicating whether the uptree branch from this node should act as an automatic choke to impose a flow rate limit.
   (The uptree branch is the one towards the terminal node; the outlet branch for a production network or the inlet branch for an injection network.)
   An automatic choke acts to control the flow of a target group nominated in item 5 by adjusting the pressure drop across the choke (see "Automatic chokes" in the ECLIPSE Technical Description). The branch must be given a VFP table number of 9999 in keyword BRANPROP so that its only pressure drop is that of the choke. Note that a terminal node cannot be selected for this purpose, as it has no uptree branch and its pressure is fixed.

   Yes  The node’s uptree branch acts as an automatic choke. A production rate target applied to the group named in item 5 will be met by adjusting the pressure loss across the choke.

   No   A production target applied to the corresponding group is met by the standard methods of group control.

   Manifold groups, and target groups for automatic chokes (item 5), may either have their rate targets set directly with GCONPROD or indirectly as a share of a higher level group’s rate target. In the latter case, that is, if they are subordinate to a higher level group having a production rate target (or a limit that becomes a rate target if violated), they should be given guide rates in keyword GCONPROD. Groups subordinate to an automatic choke’s target group should not have guide rates. Production wells subordinate to an automatic choke’s target group are not subject to guide rate group control.
The convergence tolerance and iteration limit for the calculation of the manifold group’s THP, or the choke’s pressure loss, are set in keyword NETBALAN. It may be necessary to increase the network balancing iteration limit (item 3) substantially if there are any automatic chokes.

**ECLIPSE 100**

The THP value input in item 10 of keyword WCONPROD for the first well in the group will be used as the initial guess for a manifold group’s THP in the first timestep. A suitable value entered for this quantity speeds the convergence of the first calculation.

**DEFAULT:** NO

4. Flag indicating whether gas-lift gas from the corresponding group’s subordinate wells should be added to the produced gas entering the network at this source node.

**NO** Do not include any lift gas flow; only produced gas enters the network at this node.

**YES** Add the lift gas flows of the subordinate producers to the gas flow entering the network at this node. The lift gas flow is assumed to be equal to the sum of the ALQ values of the subordinate producers, multiplied by their efficiency factors unless instructed otherwise in WEFAC item 3. The ALQ variable for the wells must therefore be defined as the rate of lift gas injection. This definition must be selected in the VFP i program if it is used to construct the VFP tables.

This item is only relevant to source nodes in a production network, which pass fluid into the network (well groups, satellite groups or Reservoir Coupling master groups). In ECLIPSE 100 they must have the same name as their corresponding groups; in ECLIPSE 300 the correspondence is defined in item 6.

Default this item if the node is not a source node in a production network.

**DEFAULT:** NO

5. Name of the group whose rate target the automatic choke attempts to match by adjusting the pressure drop across it.

For a production network node, the choke attempts to match this group’s production rate target (GCONPROD). For a water/gas injection network node in ECLIPSE 300, the choke will attempt to match the group’s water/gas injection target (GCONINJE).

This item is relevant only if item 3 is set to YES.

**ECLIPSE 100**

If the node in item 1 corresponds to a well group, default this item in ECLIPSE 100 since manifold groups must always respond to their own rate targets.

**DEFAULT:** The group of the same name as item 1, if one exists.

**ECLIPSE 300 only**

6. For a source/sink node, the name of the corresponding group.

This item is relevant only to source nodes in a production network (where fluid produced from the formation enters the network) and sink nodes in an injection network (where fluid leaves the network to be injected into the formation). The node should correspond to a well group or a satellite group. The item names the group that corresponds to the node named in item 1. The source/sink node’s inflow/outflow will be equated with the production/injection of the nominated group. (The setting of item 3 of keyword WEFAC dictates whether or not the well flows are multiplied by their efficiency factors for this purpose.)

This item is ignored if the node is not a source/sink node. It is also ignored in ECLIPSE 100 because here the source/sink nodes must have the same names as their corresponding groups.

**DEFAULT:** The group of the same name as item 1, if one exists.
The type of network to which the node belongs

**PROD**  Production network

**WINJ**  Water injection network

**GINJ**  Gas injection network

All the nodes within a given network (subordinate to the same terminal node) must have the network same type.

This item is ignored in ECLIPSE 100 because the extended network structure supports only production networks.

**DEFAULT:** **PROD**

End the data record with a slash (/).

End the set of records with a blank record containing just a slash.

See also keyword **WNETDP**.

### Example

<table>
<thead>
<tr>
<th>NODEPROP</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>-- Node</strong></td>
</tr>
<tr>
<td><strong>-- name</strong></td>
</tr>
<tr>
<td><strong>FIELD</strong></td>
</tr>
<tr>
<td><strong>GR-A1</strong></td>
</tr>
<tr>
<td><strong>GR-A2</strong></td>
</tr>
<tr>
<td><strong>GR-B1</strong></td>
</tr>
<tr>
<td><strong>GR-B2</strong></td>
</tr>
</tbody>
</table>

/
NODPCDT  Disable DPCDT option

This keyword disables the DPCDT option.

This keyword has no associated data
**NODPCO**  
**No dual porosity condensation**

In a dual porosity (rather than dual permeability) run, it is possible to eliminate the matrix cells prior to entering the linear solver. This yields a significant gain in solver speed, but there are some restrictions: for example, wells may not be completed in the matrix cells.

NODPCO suppresses this elimination, and may be used to avoid these restrictions.

This keyword has no associated data.

This keyword is ignored in ECLIPSE 100.

---

<table>
<thead>
<tr>
<th>Keywords</th>
<th>ECLIPSE 100</th>
<th>ECLIPSE 300</th>
<th>SPECIAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td></td>
<td>RUNSPEC</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>GRID</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>EDIT</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>PROPS</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>REGIONS</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>SOLUTION</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>SUMMARY</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>
**NODPPM**

**No dual porosity permeability multiplier**

In a dual porosity or dual permeability run (keywords `DUALPORO` or `DUALPERM` in the `RUNSPEC` section) the permeability values for the fracture cells are normally multiplied by the fracture porosity to yield a net bulk fracture permeability.

If the `NODPPM` keyword is used this operation is suppressed, so that you must enter net bulk permeability values directly.

This keyword has no associated data.
**NOECHO**

**Disable echoing of the input file**

The keyword causes the echo of the data input that is produced at the start of each run to be switched off from the next keyword until a subsequent ECHO keyword is encountered (or until the end of the data).

NOECHO may be used to reduce the amount of print-out from a run, or to avoid the output of large included files.

ECHO and NOECHO may be specified in any section, and any number of times in an input file.

The NOECHO keyword has no associated data.

---

```
<table>
<thead>
<tr>
<th>x</th>
<th>ECLIPSE 100</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>ECLIPSE 300</td>
</tr>
<tr>
<td></td>
<td>SPECIAL</td>
</tr>
<tr>
<td>x</td>
<td>RUNSPEC</td>
</tr>
<tr>
<td>x</td>
<td>GRID</td>
</tr>
<tr>
<td>x</td>
<td>EDIT</td>
</tr>
<tr>
<td>x</td>
<td>PROPS</td>
</tr>
<tr>
<td>x</td>
<td>REGIONS</td>
</tr>
<tr>
<td>x</td>
<td>SOLUTION</td>
</tr>
<tr>
<td>x</td>
<td>SUMMARY</td>
</tr>
<tr>
<td>x</td>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>
```
NOfREEZE  Suppress use of the FREEZEPC option

This keyword suppresses the use of the FREEZEPC option. This is no longer the default in any solution mode, so NOfREEZE is only now required for back-compatibility.

NOfREEZE contains no items.
DO NOT OUTPUT A GRID GEOMETRY FILE

Unless this keyword is specified, the program produces a file containing a definition of the grid geometry, to be used for graphical output.

This keyword has no associated data.

See also keyword GRIDFILE and "File handling in ECLIPSE" in the ECLIPSE Technical Description.
**NOHMD**

### Switch off calculation of derivatives

The keyword switches off calculation of the derivatives for the gradient option. Once a model has been set up and history matched, this allows a prediction run to be advanced without the extra cost of calculating the derivatives.

Once switched off, the calculation may not be switched on again because information from previous times is required to compute the derivatives.

The keyword should be followed by a list of names of gradient parameters to be switched off. These may be either:

- Region keywords `HMxxxxxx`
- `HMFAULTS`
- `HMMULRGT`
- `HMAQUCT` and `HMAQUFET`
- `HMROCK` and `HMROCKT`

If the list is defaulted, then calculation of all the derivatives is switched off.

In ECLIPSE 300 it is not possible to selectively switch off derivative calculations. If the NOHMD keyword is found all derivative calculations will be switched off.

The keyword should be followed by a slash (/).

### Example

Switch off parameter `HMPORVM` and all derivatives for Carter-Tracy aquifers

```
NOHMD
' HMPORVM' 'HMAQUCT' /
```
Switch off calculation of derivatives

The keyword switches off calculation of the derivatives for the gradient option. Once a model has been set up and history matched, this allows a prediction run to be advanced without the extra cost of calculating the derivatives.

Once switched off, the calculation may not be switched on again because information from previous times is required to compute the derivatives.

The keyword should be followed by a list of names of gradient parameters to be switched off. These may be either:

- Region keywords `HMxxxxxx`
- `HMFAULTS`
- `HMMULRGT`
- `HMAQUCT` and `HMAQUFET`
- `HMROCK` and `HMROCKT`

If the list is defaulted, then calculation of all the derivatives is switched off.

In ECLIPSE 300 it is not possible to selectively switch off derivative calculations. If the `NOHMO` keyword is found all derivative calculations will be switched off.

The keyword should be followed by a slash (/).

*Note:* `NOHMO` is an alias for `NOHMD`.

Example

Switch off parameter `HMPORVM` and all derivatives for Carter-Tracy aquifers

```
NOHMO
  'HMPORVM' 'HMAQUCT'
```
**NOHYKR**  
**Do not apply hysteresis to relative permeabilities**

This keyword suppresses the use of the hysteresis option in relative permeability calculations. It is now strongly recommended to control this option using the item 5 of the *EHYSTR* keyword.

If the hysteresis option is selected using the *HYST* keyword, either *HYKR* or *NOHYKR* must be entered as well.

This keyword has no associated data.
NOHYPC  Do not apply hysteresis to capillary pressure

This keyword suppresses the use of the hysteresis option in capillary pressure calculations. It is now strongly recommended to control this option using the item 5 of the EHYSTR keyword.

If the hysteresis option is selected using the HYST keyword, either HYPC or NOHYPC must be used.

This keyword has no associated data.
NOHYST  

**Enable hysteresis option**

This keyword turns off the hysteresis option. Any IMBNUM values entered in the REGIONS section are ignored. IMBNUM data results in an error if it is not being used, unless NOHYST is entered.

This keyword has no associated data.
Disables the initial index file output

This keyword requests that an initial index file describing the arrays in the INIT file should not be written. The default behavior is for the initial index file to be written.

See "File handling in ECLIPSE" in the ECLIPSE Technical Description for further information.

The keyword has no associated data.
NOLTRACE    Prevent streamline tracing for the Lagrangian Tracer Solver option

The NOLTRACE keyword contains no items. When this keyword is encountered, streamlines will not be retraced until either an LTRACE or a LAGTRUPD keyword has been read in.

Example

```
NOLTRACE
```

Keywords
NOMIX

Select no interpolation

By default, in three-phase runs, the oil and gas relative permeabilities are interpolated near the critical point to prevent un-physical discontinuities which may cause problems in solving the flow equations. NOMIX suppresses the interpolation, and may be used to obtain compatibility with other simulators, or in studies where the discontinuity is not serious. A description of this interpolation may be found in "Near critical oil and gas relative permeabilities" in the ECLIPSE Technical Description.

This keyword has no associated data.

This facility is not available in ECLIPSE 100; this keyword will be ignored.

In ECLIPSE 300, this option will be selected automatically if the ODDP relative permeability and capillary pressure hysteresis model is selected.
NOMONITO  Disable output for run-time monitoring

This keyword requests that information required by GRAF for the run-time monitoring option should not be written to the summary specification file. By default, run-time monitoring is always switched on. It can also be turned on with the MONITOR keyword.

The NOMONITO keyword has no associated data.

Example

```
NOMONITO
```
Disallow non-neighbor connections

This indicates that non-neighbor connections are **not** allowed. The default is to allow non-neighbor connections. When ECLIPSE can be run without non-neighbor connections, performance may be improved. However, many ECLIPSE features require non-neighbor connections, so this keyword should only be used when it is known that a performance improvement may be made with it.

Non-neighbor connections can be generated by several facilities. These include:

- numerical aquifers,
- faulted corner point geometry,
- dual porosity,
- the NNC-based nine-point scheme, and
- completing the circle in radial studies.
- the WARP solver, in most cases, including parallel runs (ECLIPSE 300 only).
- connections added explicitly by the user, using the NNC keyword in the GRID section.

When any of these features are present, this keyword must not be used; an error or warning will result if it is included.

The keyword has no associated data.
NORSSPEC  Disables the restart index file output

The keyword requests that a restart index file should not be written at the end of the run. By default, the restart index file will always be written. The file contains a list of the arrays written to the Restart file(s) at each restart time.

See "File handling in ECLIPSE" in the ECLIPSE Technical Description for further information.

The keyword has no associated data.
NOSIM

Turn off simulation

This keyword turns off timestep simulation in the SCHEDULE section, but the simulator will continue reading and checking the data. This is intended primarily to verify the validity of the keyword syntax prior to submitting a large simulation run.

The keyword has no associated data.
NOWARN

Suppress ECLIPSE warning messages

If this keyword is specified, all ECLIPSE warning messages are suppressed after this keyword has been read.

*Note:* General extrapolation warnings for PVT and VFP tables are suppressed by default. They are only issued if requested with the `EXTRAPMS` keyword.

Warning messages can be switched on again using the `WARN` keyword.
NOWARNEP  Suppress warning messages relating to consistency of saturation table end points

If this keyword is specified, all ECLIPSE 100 warning messages relating to consistency between saturation table end points are suppressed. Error messages are not suppressed.

The keyword has no associated data.
NPROCX

Set number of processors in the x-direction in a parallel run

This is an optional keyword associated with the `PARALLEL` keyword. When running in parallel the reservoir model is divided up in a regular areal pattern. The arrangement of processors is an \( N_{px} \) by \( N_{py} \) grid, where \( N_{px} \) is the item in `NPROCX` and \( N_{py} \) is the items in `NPROCY`.

The product of \( N_{px} \) and \( N_{py} \) must equal \( N_{procs} \), the item in the `PARALLEL` keyword.
NPROCY  Set number of processors in the y-direction in a parallel run

This is an optional keyword associated with the PARALLEL keyword. When running in parallel the reservoir model is divided up in a regular areal pattern. The arrangement of processors is an \( N_{px} \) by \( N_{py} \) grid, where \( N_{px} \) is the item in NPROCX and \( N_{py} \) is the item in NPROCY.

The product of \( N_{px} \) and \( N_{py} \) must equal \( N_{procs} \), the item in the PARALLEL keyword.
NRSOUT

The maximum number of quantities that can be output to a restart file

This keyword need only be used if ECLIPSE has given an error message requesting it.

The keyword should be followed by one item, terminated by a slash (/).

1. The maximum number of quantities that can be output at a restart file at each report step.

   DEFAULT: 3600

Note: The above number will not be the same as the number of quantities asked for in the RPTST keyword, for example. ECLIPSE adds headers and some other data for its own use that increases the size of the restart file above the number of user-defined quantities.

The data should be terminated by a slash (/).

Example

```
NRSOUT 5000/
```
NSTACK  Linear solver stack size

The data represents the size of the stack of previous search directions held by the linear solver (see "Solution of the linear equations" in the ECLIPSE Technical Description). This may be increased if the linear solver shows convergence problems. A good value for many ECLIPSE 100 problems is 10, while many compositional ECLIPSE 300 problems often require 40. Difficult problems may require a larger value. ECLIPSE prints a message advising increased NSTACK if this will increase computing efficiency. Increasing the value of NSTACK increases the memory required for a run.

The keyword should be followed by these data items, terminated by a slash (/).

1. The stack size for the ORTHOMIN or CPR linear solver.
   
   DEFAULT:

   10  ECLIPSE 100
   40  ECLIPSE 300
   20  ECLIPSE 300 using the JALS solver or CPR solver

2. Not used at present.

The data should be terminated by a slash (/).

Examples

ECLIPSE 100 only

There is no point in setting NSTACK larger than the maximum number of linear iterations (see LITMAX in keyword TUNING). The maximum possible NSTACK is printed by ECLIPSE as soon as it has computed the amount of memory required for the run.

ECLIPSE 100 only

NSTACK can only be specified in the RUNSPEC section.

Example in ECLIPSE 100 only

```plaintext
NSTACK
25/
```

ECLIPSE 300 only

The maximum number of linear iterations, specified using the CVCRIT keyword, should usually be at least the size of the stack.

ECLIPSE 300 only

NSTACK can be specified in either the RUNSPEC or the SCHEDULE section.

Example in ECLIPSE 300 only

```plaintext
NSTACK
50/
```
Net to gross thickness ratios

The keyword should be followed by one non-negative real number for every grid block in the current input box. The values specified are used to convert from gross to net thicknesses, and act as multipliers of grid block pore volumes and transmissibilities in the X and Y (or R and Theta) directions, and also on DZ for the calculation of well connection transmissibility factors. The data must be terminated by a slash (/).

Any NTG values which are not specified when the end of the GRID section is reached default to 1.0.

Note: The use of both NTG and DZNET keywords in the same run is not allowed.

Grid blocks whose pore volume is zero are treated by ECLIPSE as inactive. Since the computing time and storage space requirements of a run depend primarily on the number of active grid blocks, it is important that inactive blocks should be unambiguously identified. This is best done by setting either PORO or NTG to zero in inactive blocks. It is also possible to set pore volumes directly (array PORV in the EDIT section), or to use the ACTNUM keyword.

Note: NTG values only apply to the pore volumes by default. Pre-2007.1 releases of ECLIPSE also scaled the rock volumes by the NTG values. Using item 117 in keyword OPTIONS3 set to a nonzero value will enable the old behavior. For thermal runs both the rock volume and the pore volume should be zero for a cell to become inactive.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

Example

```
------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX  6 11 4 9 2 3 /
NTG 6*0.4 6*0.4 6*0.48 6*0.51 6*0.7 6*0.72
    6*0.4 6*0.4 6*0.48 6*0.51 6*0.7 6*0.72 /
```
**NUMRES**

### Multiple reservoirs used

This keyword specifies the number of reservoirs in the grid, if greater than one. This should be set if corner point geometry (see keywords `COORD`, `ZCORN` in the `GRID` section) is used with more than one set of coordinate lines. If more than one reservoir is used, then that number of sets of `COORD` data should be entered in the `GRID` section using the keyword `RESVNUM`, and keyword `COORDSYS` should be used to specify the bounds of each reservoir.

The data should be terminated with a slash (/).

**DEFAULT:** 1

**Example**

```
NUMRES
  3 /
```
Number of iterations to update well targets

This keyword sets the number of non-linear iterations for which a full hierarchical production/injection system solution is carried out. During any subsequent non-linear iterations of the timestep, the well rate targets will remain unchanged to allow convergence.

When any form of group control option is being used (for example, keyword GCONPROD), the flow targets of certain wells are directly influenced by the behavior of other wells in the field. For example, the target rates of producers under group control must allow for the production from wells that are under other modes of control within the group. Also injection wells performing reinjection or voidage replacement have flow targets that depend on the behavior of the production wells.

For the group targets to be met exactly, the flow targets of the wells under group control must be recalculated at every Newton iteration. But when the well flow targets change at each iteration, the convergence rate of the iterations is reduced. This is because the Jacobian matrix calculation does not include the terms representing the mutual dependency of the well rates.

As a compromise, the well flow targets are updated in the first NUPCOL Newton iterations of each timestep, and are kept constant for any subsequent iterations. Thus the group/field flow targets are met exactly if the timestep converges within NUPCOL iterations. Any changes in the solution after the first NUPCOL iterations may cause the group/field flow rates to drift slightly away from their targets. In general, larger values of NUPCOL result in the group/field flow targets being met more accurately, but perhaps at the expense of requiring more Newton iterations to achieve convergence.

The value of NUPCOL is initialized to 3 in ECLIPSE 100 and 2 in ECLIPSE 300. This keyword may be used to change the value of this quantity at any time in the simulation.

The keyword should be followed by an integer denoting the last non-linear iteration for which well rate targets are recalculated. The integer should be followed by a slash.

See also keyword GCONTOL.

Notes

- If the NUPCOL keyword is used in a restart run it must be specified after the restart time, or it will be ignored and the value from the base run will be used instead.

- NUPCOL is also used in other situations where the control mode, or some other quantity, in the well needs to be 'frozen' during the Newton iterations. Some examples are WELDRAW, during gas lift optimization (LIFTOPT), solving networks (NETBALAN) and during prioritization (see keywords PRIORITY and PRORDER for example).

Example

```
NUPCOL 4 /
```
NWATREM  Removes water from a node in the extended network

This keyword can be used in the extended network (constructed with keywords BRANPROP and NODENODEPROP, with keyword NETWORK present in the RUNSPEC section) to remove water from the network at a specified node. The amount of water to be removed can be expressed as either a constant rate or a fraction of the rate flowing through that node. The removal of water affects the pressure loss in the downstream network branches as their VFP tables are looked up with a smaller water fraction. However, this is the only effect that it has. It does not affect the reported group production rates or the operation of any group water rate targets/limits, reinjection or group economic limits; the group water rate still represents the rate of water produced from the formation.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Node name
2. Maximum rate of water removal from the node
   The specified rate of water is removed, subject to a maximum fraction of the node’s water flow entered in item 3.
   UNITS: sm³/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm³/day (PVT-M)
   DEFAULT: Infinity
3. Maximum fraction of the node’s water flow to be removed
   The specified fraction of the node’s water flow is removed, subject to a maximum rate entered in item 2.
   DEFAULT: 1.0

Note: The defaults in items 2 and 3 only apply to nodes that have been named in item 1 of this keyword. All other nodes retain their initial values of 0.0 for both quantities, so that no water is removed from them.

CAUTION: Since water removal only affects the pressure losses in the downstream branches, an error is reported if the node in item 1 has no outflow branch.

Examples

Example 1
Remove 20,000 units/day of water from the node. Item 3 defaults to 1.0 to ensure that the water removal rate does not exceed the flow rate of water through the node.

```plaintext
NWATREM
  NODE-5    20000   /
/  
```

Keywords

1472
Example 2

Remove half of the water that flows through the node. Item 2 defaults to infinity so that there is no other limit on the removal rate.

```
NWATREM
  NODE-5  1*  0.5  /
```
**NXFIN**

**Number of local cells in each global cell of an LGR in X direction**

If a local grid refinement covers more than one global cell in the x-direction, **NXFIN** can be used to dictate how many local cells each of the global cells is divided into. The keyword should be placed after the keyword **CARFIN** introducing the local grid, and before the local grid data is terminated with **ENDFIN**. **NXFIN** does not apply to radial refinements introduced with **RADFIN**.

**NXFIN** should be followed by \(I_2-I_1+1\) values terminated with a slash (/), where \(I_1\) and \(I_2\) are the I-coordinates defining the box of global grid cells to be refined (items 2 and 3 in keyword **CARFIN**). The number of values is thus the number of global cells of the refinement counted along the X-direction. The values represent the number of local cell divisions, counted along the X-direction, in each of the global cells. The sum of the values must, of course, be equal to \(NX\) set in item 8 of **CARFIN**.

In the absence of this keyword, the global cells are refined to contain equal numbers of local cells in the x-direction.

Local grid refinement is described in "Local grid refinement and coarsening" in the *ECLIPSE Technical Description*.

**Example**

```plaintext
CARFIN
-- NAME   I1 I2  J1 J2 K1 K2 NX NY NZ
LGR2   3 4   1 2  5 6  4 2  4 /
--Two global cells in x-direction to become 4 local cells
--with three local cells in first global, 1 in second
NXFIN
  3 1 /
```

**Keywords**
NYFIN  Number of local cells in each global cell of an LGR in Y direction

If a local grid refinement covers more than one global cell in the y-direction, NYFIN can be used to dictate how many local cells each of the global cells is divided into. The keyword should be placed after the keyword CARFIN introducing the local grid, and before the local grid data is terminated with ENDFIN. NYFIN does not apply to radial local grids introduced with RADFIN.

NYFIN should be followed by J2-J1+1 values terminated with a slash (/), where J1 and J2 are the J-coordinates defining the box of global grid cells to be refined (items 4 and 5 in keyword CARFIN). The number of values is thus the number of global cells of the refinement counted along the Y-direction. The values represent the number of local cell divisions, counted along the Y-direction, in each of the global cells. The sum of the values must, of course, be equal to NY set in item 9 of CARFIN.

In the absence of this keyword, the global cells will be refined to contain equal numbers of local cells in the y-direction.

The form of the input is analogous to that of NXFIN.
NZFIN

Number of local cells in each global cell of an LGR in Z direction

If a local grid refinement covers more than one global cell in the z-direction, NZFIN can be used to dictate how many local cells each of the global cells is divided into. The keyword should be placed after the keyword CARFIN or RADFIN introducing the local grid, and before the local grid data is terminated with ENDFIN.

NZFIN should be followed by K2-K1+1 values terminated with a slash (/), where K1 and K2 are the K-coordinates defining the box of global grid cells to be refined (items 4 and 5 in either keyword CARFIN or RADFIN). The number of values is thus the number of global cells of the refinement counted along the Z-direction. The values represent the number of local cell divisions, counted along the Z-direction, in each of the global cells. The sum of the values must, of course, be equal to NZ set in item 10 of CARFIN, or in item 8 of RADFIN.

In the absence of this keyword, the global cells will be refined to contain equal numbers of local cells in the z-direction.

The form of the input is analogous to that of NXFIN.
This section contains an alphabetic listing of keywords starting with the letter O.
Request ODD3P three-phase relative permeability and capillary pressure model

This keyword, which has no associated data, is used to specify that the three-phase relative permeability and capillary pressure values are to be calculated using the ODD3P method. This is a fully coupled model which incorporates hysteresis and miscibility effects for both relative permeability and capillary pressure. The ODD3P keyword should only be used in three-phase runs.

For the oil phase, the oil-water and oil-gas tables are specified using the SOF3 keyword. For the water phase, the water-oil and water-gas tables must be entered using the SWF3 keyword (rather than SWFN). For the gas phase, the gas-oil and gas-water tables must be entered using the SGF3 keyword (rather than SGFN).

The ODD3P method can be used with endpoint scaling. In this case the primary end-point scaling is specified using the PSORG, PSGRO, PSORW, PSWRO, PSGRW and PSWRG keywords and the hysteresis end-point scaling is specified using the HSORG, HSGRO, HSORW, HSWRO, HSGRW and HSWRG keywords.

ECLIPSE uses the default three-phase oil relative permeability model if the keywords STONE, STONE1, STONE2, or ODD3P are omitted from the PROPS section of the input data file.

See "Saturation functions" in the ECLIPSE Technical Description for a description of three-phase oil relative permeability models available in ECLIPSE.

Example

```
ODD3P
```

Keywords

| Keywords | 1478 |
Requests output of SUMMARY data in OFM format

This keyword requests post-processing of the SUMMARY output into a form that can conveniently be input into the OFM program. The process is analogous to the generation of run summaries (RUNSUM keyword), in that the SUMMARY files are post-processed at the end of the run to convert the SUMMARY output into a new format.

If the OFM keyword is present then either 3 or 4 new files will be generated:

- **ROOT_OFM.DEF** Defines the tables in the OFM data model
- **ROOT_OFM.XY** defines the areal (XY) location of either the well or completion
- **ROOT_OFM_WELL.PRD** Well data
- **ROOT_OFM_COMP.PRD** Completion data

The output is generated for a set of unique identifiers based on connection data if available, and on well data if not. In the case of connections the COFR mnemonic is the controlling mnemonic (if no COFR is present then CGFR and CWFR are alternatives). The unique identifiers take the form WELL:AA_BB_CC where AA, BB, CC are the X,Y,Z location of the connection.

It is assumed that if completion data is available, then the well aggregates, etc., will be calculated in OFM. This prevents the possibility of multiple data items carrying the same information once input into the OFM database.

If the controlling connection mnemonic is not present for a given well then the controlling mnemonic for well level data is the WOPR (and if this keyword is not present, then WGPR and WWPR are alternatives). In the case of well data the unique identifier is simply the well name.

For all completions with COFR data the output consists of the superset of all completion mnemonics (C* keywords) present in the SUMMARY section. If no COFR keyword is present and WOPR is present for a given well, the data consists of the superset of all well mnemonics (W* keywords). Note that it is possible to produce very large files in cases where the superset of mnemonics used is large combined with large numbers of connections and timesteps.

The following conversions of mnemonic names are carried over automatically to the standard OFM name:

- COFR *OIL
- CWFR *WATER
- CGFR *GAS
- CPR *PRESSURE
- WOPR *OIL
- WWPR *WATER
- WGPR *GAS
- WBHP *PRESSURE

All other mnemonics retain their keyword names.
Once the ECLIPSE run has completed and the OFM-ready files have been produced, these files can easily be imported into OFM by creating a new project and then loading data from ASCII flat files. In the load panel visible after a new project has been created, select then load the four OFM files from a run. The project will then be populated.

The X,Y location of the well will normally be referenced relative to the grid. However, if the MAPAXES keyword is present in the dataset, then an effective transform to the new MAPAXES origin is performed.

The OFM keyword has no associated data.

See also the keyword RUNSUM.

**Example**

```
OFM
```
OIL

Indicates that the run contains oil

This keyword indicates that a run contains an oil phase. The OIL keyword should be specified whenever an oil phase exists or could exist.

For ECLIPSE 300, the OIL keyword is automatically supplied for any compositional mode run.

The keyword OIL contains no items.
**OILAPI**

**Initial oil API values, for API tracking option**

The keyword should be followed by one real number for every grid block, specifying the initial oil API gravity. The data field should be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword `DIMENS`. Repeat counts may be used for repeated values (for example 115*38.9). Note that spaces must not be inserted on either side of the asterisk (*).

See also the keywords `PRESSURE`, `RS`, `RV`, `SWAT` and `SGAS` in the `SOLUTION` section.

**UNITS:** °API

**Example**

With NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword `DIMENS`:

```
OILAPI 48*39.4 192*36.1 /
```

---

**Keywords**

1482
Oil compressibility

This keyword can only be used with the THERMAL option.

OILCOMPR applies to runs where the DEADOIL keyword has been specified in the RUNSPEC section.

OILCOMPR specifies the oil compressibility and thermal expansion coefficients for each pressure table. A data record should be entered for each pressure table, each containing some or all of the following items of data:

1. The oil compressibility
   
   **UNITS:** 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)
   
   **DEFAULT:** 0.0

2. The oil expansion coefficient of temperature
   
   **UNITS:** 1/K (METRIC), 1/°R (FIELD), 1/K (LAB), 1/K (PVT-M)
   
   **DEFAULT:** 0.0

3. The oil expansion coefficient of temperature squared
   
   **UNITS:** 1/K^2 (METRIC), 1/°R^2 (FIELD), 1/K^2 (LAB), 1/K^2 (PVT-M)
   
   **DEFAULT:** 0.0

**Example**

```
OILCOMPR
-- Compressibility  therm.expansion  in Field units
--                  coefficients
5.00E-06  3.80E-04  0.0  /
```
**OILMW**

**Molecular weight of oil**

This keyword can only be used with the **THERMAL** option.

OILMW applies to runs where the **DEADOIL** keyword has been specified in the RUNSPEC section.

OILMW specifies the mean molecular weight of oil.

**UNITS:** kg/kg-M (**METRIC**), lb/lb-M (**FIELD**), gm/gm-M (**LAB**), kg/kg-M (**PVT-M**)

**DEFAULT:** none

If multiple sets of pressure tables are being used, a set of data should be entered for each pressure table. The number of pressure tables is entered as item 2 of **TABDIMS**.

**Example**

Mean molecular weight for a heavy oil

```
OILMW
600 /
```

---

*Keywords* 1484
OILSPECH

Oil specific heat

This keyword can only be used with the THERMAL option.

OILSPECH applies to runs where the DEADOIL keyword has been specified in the RUNSPEC section.

OILSPECH specifies the oil specific heat for each pressure table. A data record is entered for each pressure table, each record containing one item:

1. The oil specific heat

   UNITS: kJ/kg/K (METRIC), Btu/lb°F (FIELD), J/gm/K (LAB), kJ/kg/K (PVT-M)

   DEFAULT: 0.0

Example

```
OILSPECH
-- In Field units
  0.5
```
Oil viscosity index

This keyword is optional and can only be used with the ECLIPSE 300 Thermal option.

The keyword defines functions $f_i(x)$ for each component that are used as indices in calculating the oil phase viscosity.

The data comprises NTPVT tables (item 2 of keyword TABDIMS) of values $f_i(x)$ versus $x$ for each pressure table. Each table consists of $N_c + 1$ columns of data (described below) and is terminated by a slash (/).

Column 1  Mole fraction values $x$

The values should be between 0 and 1, and increase monotonically down the column.

Columns 2 to $N_c + 1$  The corresponding values of oil component viscosity index $f_i(x)$

The values should be between 0 and 1, and increase monotonically down the column.

In addition:

$f_i(0) = 0$ and $f_i(1) = 1$

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than $N_{PVT}$ see keyword TABDIMS).

The oil phase viscosity is calculated from the component viscosities:

$$
\mu_o = \mu_T(T) = \Pi_{i=1}^{N_c} \mu_{oi} f_i(x_i)
$$

Eq. 3.124

where

$\mu_{oi}$  are the component viscosities defined by keyword OILVISCT, OILVISCC or OILVISCf,

$x_i$  are the component mole fractions in the oil phase, and

$f_i(x_i)$  are the viscosity indices defined by this keyword.

If this keyword is not used then linear functions are used:

$f_i(x_i) = x_i$

If both the PVCO and VISCREF keywords are specified, a pressure dependence modifies the above temperature-dependent viscosities. The oil viscosity at the prevailing pressure is then calculated as follows:

$$
\mu_o = \mu_T(T) \frac{\mu_p(P)}{\mu_p(P_{ref})}
$$

Eq. 3.125

where

$\mu_T$  is the viscosity defined in equation 3.124.

$\mu_p$  is the viscosity from the PVCO keyword.
$P_{\text{ref}}$ is the reference pressure defined by the VISCREF keyword.

See "Thermal option" in the ECLIPSE Technical Description.

**Example**

```
OILVINDEX
  0.0 0.0 0.00
  0.1 1* 0.01
  0.2 1* 0.04
  0.3 1* 0.09
  0.4 1* 0.16
  0.5 1* 0.25
  0.6 1* 0.36
  0.7 1* 0.49
  0.8 1* 0.64
  0.9 1* 0.81
  1.0 1.0 1.00
/
```
Oil viscosity-temperature correlation

This keyword can only be used with the THERMAL option. It can be used with the default LIVEOIL option, but not with the DEADOIL option.

OILVISCC can be used to specify the temperature dependence of the oil component viscosities using one of four standard correlations:

ASTM: \[ \log_{10}(\mu_{oi} + A) = B \cdot T^{-C} \]  
Eq. 3.126

Andrade: \[ \log_{10}(\mu_{oi}) = A + B / T \]  
Eq. 3.127

Vogel: \[ \log_{10}(\mu_{oi}) = A + B / (T + C) \]  
Eq. 3.128

logarithmic: \[ \log_{10}(\mu_{oi}) = A + B \cdot \log_{10}(T) \]  
Eq. 3.129

where \( \mu_{oi} \) is the viscosity of oil component \( i \) (cP) and \( T \) is the temperature.

UNITS: K (METRIC), °R (FIELD), K (LAB), K (PVT-M)

The keyword can be used in two different ways:

- as a correlation, in which case values of \( T \) and \( \mu_{oi}(T) \) are entered and the simulator calculates the coefficients \( A, B \) and (where applicable) \( C \)

- as a formula, in which case the coefficients \( A, B \) and (where applicable) \( C \) should be entered.

The keyword is followed by sets of records. Each set contains one record specifying the correlation name and type, followed by either two or three records that define the coefficients. The set of records is terminated with a slash.

To use the ASTM correlation four records are required:

1. ASTM CORRELATION
2. \( N_c +1 \) values are entered: a temperature \( T_1 \) and \( N_c \) viscosities \( \mu_{oi}(T_1) \), one for each hydrocarbon component.
3. \( N_c +1 \) values are entered: a temperature \( T_2 \) and \( N_c \) viscosities \( \mu_{oi}(T_2) \), one for each hydrocarbon component.
4. \( N_c \) viscosities \( \mu_{oi}(\infty) \) are entered, giving the limiting viscosity for each hydrocarbon component as the temperature tends to infinity.

**Note:**

\( \mu_{oi}(T_1) + 1 > \mu_{oi}(\infty) \) and \( \mu_{oi}(T_2) + 1 > \mu_{oi}(\infty) \)

Alternatively, the coefficients can be specified:

1. ASTM FORMULA
2. \( N_c \) values are entered for the coefficient \( A \), one for each hydrocarbon component.
3. \( N_c \) values are entered for the coefficient \( B \), one for each hydrocarbon component.
4. \( N_c \) values are entered for the coefficient \( C \), one for each hydrocarbon component.

**Note:** \( A < 1, B > 0 \) and \( C > 0 \)

To use the Andrade correlation three records are required:

1. **ANDRADE CORRELATION**
2. \( N_c \) +1 values are entered: a temperature \( T_1 \) and \( N_c \) viscosities \( \mu_{oi}(T_1) \), one for each hydrocarbon component.
3. \( N_c \) viscosities \( \mu_{oi}(\infty) \) are entered, giving the limiting viscosity for each hydrocarbon component as the temperature tends to infinity.

Alternatively, the coefficients can be specified:

1. **ANDRADE FORMULA**
2. \( N_c \) values are entered for the coefficient \( A \), one for each hydrocarbon component.
3. \( N_c \) values are entered for the coefficient \( B \), one for each hydrocarbon component.

To use the Vogel correlation four records are required:

1. **VOGEL CORRELATION**
2. \( N_c \) +1 values are entered: a temperature \( T_1 \) and \( N_c \) viscosities \( \mu_{oi}(T_1) \), one for each hydrocarbon component.
3. \( N_c \) +1 values are entered: a temperature \( T_2 \) and \( N_c \) viscosities \( \mu_{oi}(T_2) \), one for each hydrocarbon component.
4. \( N_c \) viscosities \( \mu_{oi}(\infty) \) are entered, giving the limiting viscosity for each hydrocarbon component as the temperature tends to infinity.

Alternatively, the coefficients can be specified:

1. **VOGEL FORMULA**
2. \( N_c \) values are entered for the coefficient \( A \), one for each hydrocarbon component.
3. \( N_c \) values are entered for the coefficient \( B \), one for each hydrocarbon component.
4. \( N_c \) values are entered for the coefficient \( C \), one for each hydrocarbon component.

To use the logarithmic correlation the following records are required:

1. **LOG CORRELATION**
2. \( N_c \) +1 values are entered: a temperature \( T_1 \) and \( N_c \) viscosities \( \mu_{oi}(T_1) \), one for each hydrocarbon component.
3. \( N_c \) +1 values are entered: a temperature \( T_2 \) and \( N_c \) viscosities \( \mu_{oi}(T_2) \), one for each hydrocarbon component.

Alternatively, the coefficients can be specified:

---

**Keywords**

1489
1. **LOG FORMULA**

2. \( N_c \) values are entered for the coefficient \( A \), one for each hydrocarbon component.

3. \( N_c \) values are entered for the coefficient \( B \), one for each hydrocarbon component.

If different formulae are required for different components, then the set of records can be repeated. If a formula does not apply to a component, the data should be defaulted for that component (by entering 1*, for example). If no data is entered for a component, the simulator will use viscosities from the `OILVISCF` or `OILVISCT` keyword.

The set of records is ended by a blank record, containing only a slash.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as the 9th parameter of `TABDIMS`.

The oil phase viscosity \( \mu_o \) is calculated from:

\[
\mu_o = \mu_T(T) = \Pi_{i=1}^{N_c} \mu_i f_i(x_i)
\]

Eq. 3.130

where

- \( \mu_i \) are the component viscosities,
- \( x_i \) are the component mole fractions in the oil phase, and
- \( f_i(x_i) \) are the viscosity indices defined by the `OILVINDX` keyword.

If both the `PVCO` and `VISCREF` keywords are specified, a pressure dependence modifies the above temperature dependent viscosities. The oil viscosity at the prevailing pressure is then calculated as follows:

\[
\mu_o = \mu_T(T) \frac{\mu_p(P)}{\mu_p(P_{\text{ref}})}
\]

Eq. 3.131

where

- \( \mu_T \) is the viscosity from this keyword.
- \( \mu_p \) is the viscosity from the `PVCO` keyword.
- \( P_{\text{ref}} \) is the reference pressure defined by the `VISCREF` keyword.

### Examples

#### Example 1

In this example there is one reservoir EoS region.

```
OILVISC
--
-- EoS region 1
--
ASTM CORRELATION
--TEMP VISC1 VISC2 VISC3
```
Example 2

In this example there are two reservoir EoS regions.

```
OILVISCC
--
-- EoS region 1
--
ASTM CORRELATION
--TEMP VISC1 VISC2 VISC3
 100  1* 700.0 1200.0
 200  1* 70.0 120.0
  0.0  0.1 1.0 /
LOG FORMULA
-- VISC1 VISC2-3
  9.64  2*
 2.0E10  2* /
--
-- EoS region 2
--
ASTM CORRELATION
--TEMP VISC1 VISC2 VISC3
 100  50.0 700.0 1200.0
 200  5.0 70.0 120.0
  0.0  0.1 1.0 /
```

OILVISCF

Oil viscosity function

This keyword can only be used with the THERMAL option. It can be used with the thermal default LIVEOIL option, but not the DEADOIL option.

The data comprises two constants for each oil component that determine a function to calculate oil component viscosity versus temperature. For each oil component, the oil viscosity is fitted according to the equation \( \mu_o = A \cdot T^B \) where \( T \) is the temperature.

(\( K \text{ (METRIC)}, ^\circ R \text{ (FIELD)}, K \text{ (LAB)}, K \text{ (PVT-M)} \)).

The keyword is followed by two records, each containing \( N_c \) columns of data described below. The set of records is ended by a blank record, containing only a slash.

1. \( A \) coefficients, a set of \( N_c \) values is entered, one for each hydrocarbon component
   
   UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

2. \( B \) coefficients, a set of \( N_c \) values is entered, one for each hydrocarbon component
   
   UNITS: dimensionless

The oil phase viscosity is calculated from the component viscosities:

\[
\mu_o = \mu_T(T) = \prod_{i=1}^{N_c} \mu_{oi}(x_i)
\]

Eq. 3.132

where

\( \mu_{oi} \) are the component viscosities,

\( x_i \) are the component mole fractions in the oil phase, and

\( f_i(x_i) \) are the viscosity indices defined by the OILVINDEX keyword.

If both the PVCO and VISCREF keywords are specified, a pressure dependence modifies the above temperature dependent viscosities. The oil viscosity at the prevailing pressure is then calculated as follows:

\[
\mu_o = \mu_T(T) \frac{\mu_p(P)}{\mu_p(P_{\text{ref}})}
\]

Eq. 3.133

where

\( \mu_T \) Viscosity from this keyword.

\( \mu_p \) Viscosity from the PVCO keyword

\( P_{\text{ref}} \) Reference pressure defined by the VISCREF keyword.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of TABDIMS.
Example

OILVISC
-- LIGHT COMP LIGHTER COMP
4.49E9 20.0E9
-3.4 -3.4

/
Oil viscosity versus temperature data

This keyword can be used with either the blackoil Temperature option or the compositional Thermal option to specify the temperature dependence of the oil phase viscosity. The way the keyword is used is discussed separately for each option in the next two sections, since there are some slight differences.

The interpolation method for the OILVISCT data can be specified with the OILVTIM keyword.

Temperature option

The Temperature option can be used in either ECLIPSE 100 or ECLIPSE 300; see keyword TEMP. When the OILVISCT keyword is used with the temperature option the VISCREF keyword should be specified to supply the reference conditions for the oil viscosity.

The data comprises NTPVT tables (item 2 of keyword TABDIMS) of oil component viscosities versus temperature for each pressure table. Each table consists of two columns of data (described below), and is terminated by a slash (/).

Column

1. Temperature values.
   The values should increase monotonically down the column.
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

2. The corresponding values of oil viscosity.
   The values should decrease or stay level down the column.
   UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)
   There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see keyword TABDIMS).

The values of oil viscosity in the OILVISCT keyword are assumed to be at the reference pressure and Rs value given in the VISCREF keyword. The oil viscosity at the prevailing pressure and Rs is calculated as follows:

\[
\mu_o = \mu_T(T) \frac{\mu_p(P, R_s)}{\mu_p(P_{ref}, R_{s\ ref})}
\]

Eq. 3.134

where

\( \mu_T \) Viscosity from the OILVISCT keyword.

\( \mu_p \) Viscosity from the PVTO (or PVDO) keyword.

\( P_{ref} \) Reference pressure from the VISCREF keyword.

\( R_{s\ ref} \) Reference Rs from the VISCREF keyword.

See "Temperature option" in the ECLIPSE Technical Description.
**ECLIPSE 100**

Note: ECLIPSE 100 linearly interpolates the reciprocal of the oil viscosities between data points, rather than the values themselves.

**ECLIPSE 300**

Note: The interpolation method used to obtain oil viscosities between data points can be set with the OILVTIM keyword. By default oil viscosities are linearly interpolated between data points.

**Example**

With $N_{TPVT}=1$ and $N_{PPVT} \geq 5$

<table>
<thead>
<tr>
<th>OILVISCT</th>
<th>10</th>
<th>60</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>10</td>
<td></td>
</tr>
<tr>
<td>150</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>200</td>
<td>0.8</td>
<td></td>
</tr>
<tr>
<td>250</td>
<td>0.5</td>
<td></td>
</tr>
<tr>
<td>/</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**ECLIPSE 300 Thermal option**

The Thermal option can only be used in ECLIPSE 300; see keyword THERMAL. Either this keyword or OILVISCC or OILVISCF or THANALV should be specified to determine the oil viscosity. Keyword VISCREF is optional and is only needed if a pressure dependence is required, as discussed below.

The data comprises $N_{TPVT}$ tables (item 2 of keyword TABDIMS) of oil component viscosities versus temperature for each pressure table. Each table consists of $N_c + 1$ columns of data (described below), and is terminated by a slash (/).

Column 1  Temperature values.
The values should increase monotonically down the column.
UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

Columns 2 to $N_c + 1$  The corresponding values of oil component viscosity.
The values should decrease or stay unchanged down the column.
UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than $N_{PPVT}$ (see keyword TABDIMS).

The oil phase viscosity is calculated from the component viscosities:

$$
\mu_o = \mu_T(T) = \Pi_{i=1}^{N_c} \mu_{oi} f_i(x_i)
$$

Eq. 3.135

where

$\mu_{oi}$ are the component viscosities

$x_i$ are the component mole fractions in the oil phase

**Keywords**

1495
\( f_i(x_i) \) are the viscosity indices defined by the \texttt{OILVINDEX} keyword.

For temperature values outside the interval specified, constant value extrapolation is performed.

If both the \texttt{PVCO} and \texttt{VISCREF} keywords are specified, a pressure dependence modifies the temperature dependent viscosities described above. The oil viscosity at the prevailing pressure is then calculated as follows:

\[
\mu_o = \mu_T(T) \frac{\mu_p(P)}{\mu_p(P_{ref})}
\]

Eq. 3.136

where

- \( \mu_T \) Viscosity from this keyword.
- \( \mu_p \) Viscosity from the \texttt{PVCO} keyword.
- \( P_{ref} \) The reference pressure defined by the \texttt{VISCREF} keyword.

See "Thermal option" in the \textit{ECLIPSE Technical Description}.

\textbf{Note:} Oil viscosity in the \texttt{DEADOIL} option cannot be lower than 0.1 cP. It will be reset to 0.1 cP.

\textbf{Example}

![Example](example.png)
Oil viscosity temperature interpolation method

This keyword specifies the interpolation method used for the OILVISCT table which by default is interpolated linearly for a given temperature. However, for heavy oil the viscosity may depend exponentially on the temperature and an interpolation on the logarithm of the viscosity may be more appropriate.

The keyword contains an item of data and should be terminated with a slash (/)

1. Interpolation method

   LIN Linearly interpolate the viscosity with respect to the temperature.

   LINLOG Linearly interpolate the logarithm of the viscosity with respect to the temperature and then take the exponential.

   DEFAULT: LIN

Example

```
OILVTIM
LINLOG /
```
OLDEQUIL

Perform equilibration using the pre-2001A method

Two changes were made to the ECLIPSE 300 equilibration for the 2001A release. The behavior was changed to be the same as that of ECLIPSE 100. The use of the OLDEQUIL keyword implies the pre-2001A behavior:

1. The Equilibration is performed using \( N \) horizontal blocks where \( N \) is item 9 of the \texttt{EQUIL} keyword.

2. The sign of the capillary pressure (4\textsuperscript{th} item of the \texttt{EQUIL} keyword) at the water-oil contact is reversed. A positive value now implies \( P_{\text{oil}} > P_{\text{water}} \) at the contact.
OLDTRAN Specifies block center transmissibilities

The use of OLDTRAN implies that transmissibilities are calculated from cell $\text{DX}$ (or $\text{DR}$), $\text{DY}$ (or $\text{DTHETA}$) and $\text{DZ}$ values, without reference to corner point positions. This does not permit the automatic calculation of geological fault non-neighbor transmissibilities, but allows back-compatibility with existing simulation codes using block center transmissibility calculations.

OLDTRAN has no associated data.

This is the default type of transmissibility calculation for grids specified using the $\text{DX}$ (or $\text{DR}$), $\text{DY}$ (or $\text{DTHETA}$) and $\text{DZ}$ keywords.

(See also keywords NEWTRAN and OLDTRANR.)
OLDTRANR  Specifies alternative block center transmissibilities

The use of OLDTRANR implies that transmissibilities are calculated from cell $D_X$ (or $D_R$), $D_Y$ (or $D_THETA$) and $D_Z$ values, without reference to corner point positions. This does not permit the automatic calculation of geological fault non-neighbor transmissibilities, but allows back-compatibility with existing simulation codes using block center transmissibility calculations.

See also the keywords OLDTRAN and NEWTRAN.

The calculation of transmissibility when OLDTRANR is specified is slightly different from the calculation used with OLDTRAN. See “Transmissibility calculations” in the ECLIPSE Technical Description for a description of the formulae used for Cartesian calculations.

OLDTRANR has no associated data.
Overrides default $\Omega_a$ values

In compositional runs, the program supplies a default value of the equation of state $\Omega_a$ coefficient. However, should you wish to modify this value, the OMEGAA keyword may be used. Note that only small variations around the default value are advisable and that the default value will only be modified if the user defined value is larger than 0. A separate value may be specified for each component.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

DEFAULT: Peng-Robinson: 0.457235529, RK, SRK and ZJ: 0.4274802

Example

```
OMEGAA
4*0.457 3*0.461 0.462 /
```

Keywords

1501
OMEGAADΕ  Overrides default $\Omega_a$ values for detailed composition

In compositional runs, the program supplies a default value of the equation of state $\Omega_a$ coefficient. However, should you wish to modify this value for the detailed components in a lumped simulation, the OMEGAADΕ keyword may be used. Note that only small variations around the default value are advisable and that the default value will only be modified if the user defined value is larger than 0. A separate value may be specified for each component.

Lumped simulations are enabled with the LUMPDIMS keyword in the RUNSPEC section; further details can be found in "Delumping and lumping" in the ECLIPSE Technical Description.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

DEFAULT: Peng-Robinson: 0.457235529, RK, SRK and ZJ: 0.4274802

Example

```
OMEGAADΕ
4*0.457 3*0.461 0.462 /
```

Keywords

[Table of Keywords]

1502
OMEGAAS | Overrides default surface $\Omega_a$ values

This keyword allows a different $\Omega_a$ value to be used for the production system (on the surface). The default values are those supplied with the OMEGAA keyword, or, if OMEGAA was not entered, the default equation of state $\Omega_a$ value. The format of the keyword is exactly the same as that of OMEGAA.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

DEFAULT: Peng-Robinson: 0.4572355, RK, SRK and ZJ: 0.4274802

Example

```
OMEGAAS
4*0.457 3*0.461 0.462 /
```

Keywords

ECLIPSE Reference Manual

1503
OMEGAASD  **Overrides default surface \( \Omega_a \) values for detailed compositions**

This keyword allows a different \( \Omega_a \) value to be used for the production system (on the surface) in a lumped simulation. The default values are those supplied with the **OMEGAAD** keyword, or, if **OMEGAA** was not entered, the default equation of state \( \Omega_a \) value. The format of the keyword is exactly the same as that of **OMEGAAD**.

Lumped simulations are enabled with the **LUMPDIMS** keyword in the **RUNSPEC** section; further details can be found in "Delumping and lumping" in the *ECLIPSE Technical Description*.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword **TABDIMS**. The use of multiple EoS regions is discussed in "Equations of state regions" in the *ECLIPSE Technical Description*. DEFAULT: Peng-Robinson: 0.4572355, RK, SRK and ZJ: 0.4274802

**Example**

```
OMEGAASD  4*0.457 3*0.461 0.462 /
```

**Keywords**
OMEGAB

Overrides default $\Omega_b$ values

In compositional runs, the program supplies a default value of the equation of state $\Omega_b$ coefficient. However, as in the case of $\Omega_a$, this keyword enables you to modify the value of this coefficient. As for $\Omega_a$, only a small variation is advisable and the default value will only be modified if the user defined value is larger than 0. A separate value may be specified for each component.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword `TABDIMS`. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

DEFAULT: Peng-Robinson: 0.077796074, RK, SRK and ZJ: 0.08664035

Example

```plaintext
OMEGAB 0.0777 2*0.078 0.07677 /
```
OMEGABDE  Overrides default $\Omega_b$ values for detailed compositions

In compositional runs, the program supplies a default value of the equation of state $\Omega_b$ coefficient. However, as in the case of $\Omega_a$, this keyword enables you to modify the value of this coefficient. As for $\Omega_a$, only a small variation is advisable and the default value will only be modified if the user defined value is larger than 0. A separate value may be specified for each detailed component in a lumped simulation.

Lumped simulations are enabled with the LUMPDIMS keyword in the RUNSPEC section; further details can be found in “Delumping and lumping” in the ECLIPSE Technical Description.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

DEFAULT: Peng-Robinson: 0.077796074, RK, SRK and ZJ: 0.08664035

**Example**

```plaintext
OMEGABDE 0.0777 2*0.078 0.07677 /
```

**Keywords**

<table>
<thead>
<tr>
<th>x</th>
<th>ECLIPSE 100</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>ECLIPSE 300</td>
</tr>
<tr>
<td>---</td>
<td>-------------</td>
</tr>
<tr>
<td>x</td>
<td>SPECIAL</td>
</tr>
<tr>
<td></td>
<td>RUNSPEC</td>
</tr>
<tr>
<td></td>
<td>GRID</td>
</tr>
<tr>
<td></td>
<td>EDIT</td>
</tr>
<tr>
<td>x</td>
<td>PROPS</td>
</tr>
<tr>
<td></td>
<td>REGIONS</td>
</tr>
<tr>
<td></td>
<td>SOLUTION</td>
</tr>
<tr>
<td></td>
<td>SUMMARY</td>
</tr>
<tr>
<td></td>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>

1506
OMEGABS  Overrides default surface $\Omega_b$ values

This keyword allows a different $\Omega_b$ value to be used for the production system (on the surface). The default values are those supplied with the OMEGAB keyword, or, if OMEGAB was not entered, the default equation of state $\Omega_b$ value.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

DEFAULT: Peng-Robinson: 0.0779607, RK, SRK and ZJ: 0.08664035

Example

```
OMEGABS
  0.0777 2*0.078 0.07677 /
```

Keywords

<table>
<thead>
<tr>
<th>ECLIPSE 100</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
</tr>
<tr>
<td>ECLIPSE 300</td>
</tr>
<tr>
<td>SPECIAL</td>
</tr>
<tr>
<td>RUNSPEC</td>
</tr>
<tr>
<td>GRID</td>
</tr>
<tr>
<td>EDIT</td>
</tr>
<tr>
<td>x</td>
</tr>
<tr>
<td>PROPS</td>
</tr>
<tr>
<td>REGIONS</td>
</tr>
<tr>
<td>SOLUTION</td>
</tr>
<tr>
<td>SUMMARY</td>
</tr>
<tr>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>
OMEGABSD  Overrides default surface $\Omega_b$ values for detailed compositions

This keyword allows a different $\Omega_b$ value to be used for the production system (on the surface) in a lumped simulation. The default values are those supplied with the OMEGABDE keyword, or, if OMEGAB was not entered, the default equation of state $\Omega_b$ value.

Lumped simulations are enabled with the LUMPDIMS keyword in the RUNSPEC section; further details can be found in "Delumping and lumping" in the ECLIPSE Technical Description.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

DEFAULT: Peng-Robinson: 0.07779607, RK, SRK and ZJ: 0.08664035

Example

```
OMEGABSD
0.0777 2*0.078 0.07677 /
```

Keywords

1508
OPERATE  Performs operations on arrays

This keyword is used to perform arithmetic operations on property arrays.

The keyword may be followed by any number of records, each of which is terminated by a slash (/). The data is terminated by a blank record, containing only a slash.

Each record consists of:

1. The result array.
   
   Items 2-7 may be used to redefine the input box for this and subsequent operations within the current keyword. If items 2-7 are not defined, they default to the values which were used for the previous operation within the current keyword. For the first operation in the keyword, the box defaults to the values set by the most recent BOX or ENDBOX keyword. If there is no preceding BOX or ENDBOX in the current section, the box is taken to include the entire reservoir.

2. First block to be modified on the X axis (IX1)
3. Last block to be modified on the X axis (IX2)
4. First block to be modified on the Y axis (JY1)
5. Last block to be modified on the Y axis (JY2)
6. First block to be modified on the Z axis (KZ)
7. Last block to be modified on the Z axis (KZ2)

The data must satisfy:

\[ 1 \leq IX1 \leq IX2 \leq NDX \]
\[ 1 \leq JY1 \leq JY2 \leq NDY \]
\[ 1 \leq KZ1 \leq KZ2 \leq NDZ, \]

where NDX, NDY, NDZ are the limits in the current BOX.

8. The type of operation. The table below describes valid operations. Let \( X_i \) denote the \( i \) th element of the argument array and \( R_i \) the \( i \) th element of the result array. The Greek letters \( \alpha \) and \( \beta \) are scalar parameters. Examples for valid arrays are given later. All operations are performed with floating point arithmetic. If the result is an integer array, the real value is rounded to the nearest integer.

<table>
<thead>
<tr>
<th>Result</th>
<th>item 8</th>
<th>item 9</th>
<th>item 10</th>
<th>item 11</th>
</tr>
</thead>
<tbody>
<tr>
<td>( R_i = \alpha X_i + \beta )</td>
<td>‘MULTA’</td>
<td>( X )</td>
<td>( \alpha )</td>
<td>( \beta )</td>
</tr>
<tr>
<td>( R_i = R_i + \alpha X_i \beta )</td>
<td>‘POLY’</td>
<td>( X )</td>
<td>( \alpha )</td>
<td>( \beta )</td>
</tr>
<tr>
<td>( R_i = 10^{(\alpha + \beta X_i)} )</td>
<td>‘SLOG’</td>
<td>( X )</td>
<td>( \alpha )</td>
<td>( \beta )</td>
</tr>
<tr>
<td>( R_i = \log 10(X_i) )</td>
<td>‘LOG10’</td>
<td>( X )</td>
<td>n/a</td>
<td>n/a</td>
</tr>
<tr>
<td>( R_i = \ln (X_i) )</td>
<td>‘LOGE’</td>
<td>( X )</td>
<td>n/a</td>
<td>n/a</td>
</tr>
<tr>
<td>( R_i = \frac{1}{X_i} )</td>
<td>‘INV’</td>
<td>( X )</td>
<td>n/a</td>
<td>n/a</td>
</tr>
</tbody>
</table>
### Table 3.21: Valid operation types

<table>
<thead>
<tr>
<th>Result</th>
<th>item 8</th>
<th>item 9</th>
<th>item 10</th>
<th>item 11</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R_i = \alpha X_i$</td>
<td>‘MULTX’</td>
<td>$X$</td>
<td>$\alpha$</td>
<td>n/a</td>
</tr>
<tr>
<td>$R_i = \alpha + X_i$</td>
<td>‘ADDX’</td>
<td>$X$</td>
<td>$\alpha$</td>
<td>n/a</td>
</tr>
<tr>
<td>$R_i = X_i$</td>
<td>‘COPY’</td>
<td>$X$</td>
<td>n/a</td>
<td>n/a</td>
</tr>
<tr>
<td>$R_i = \min(\alpha , X_i)$</td>
<td>‘MAXLIM’</td>
<td>$X$</td>
<td>$\alpha$</td>
<td>n/a</td>
</tr>
<tr>
<td>$R_i = \max(\alpha , X_i)$</td>
<td>‘MINLIM’</td>
<td>$X$</td>
<td>$\alpha$</td>
<td>n/a</td>
</tr>
<tr>
<td>$R_i = \alpha X_i^\beta$</td>
<td>‘MULTP’</td>
<td>$X$</td>
<td>$\alpha$</td>
<td>$\beta$</td>
</tr>
<tr>
<td>$R_i = \text{abs}(X_i)$</td>
<td>‘ABS’</td>
<td>$X$</td>
<td>n/a</td>
<td>n/a</td>
</tr>
<tr>
<td>$R_i = X_iR_i$</td>
<td>‘MULTIPLY’</td>
<td>$X$</td>
<td>n/a</td>
<td>n/a</td>
</tr>
</tbody>
</table>

9. Array parameter depending on item 8. Must be supplied, no default.
10. Scalar parameter depending on item 8. Must be supplied if required, no default.
11. Scalar parameter depending on item 8. Must be supplied if required, no default.

### Legal arrays

#### GRID section arrays

- DX (DR)
- DY (DTHETA)
- DZ
- PERMX (PERMR)
- PERMY (PERMTHT)
- PERMZ
- MULTX (MULTR)
- MULTY (MULTTHT)
- MULTZ
- PORO
- NTG
- DZNET
- TOPS
- DIFFMX (DIFFMR) DIFFMY (DIFFMTHT) DIFFMZ

*ECLIPSE 300 only*

- MIDS

#### EDIT section arrays

- PORV
- DEPTH
- TRANX (TRANR) TRANY (TRANTHT) TRANZ
- DIFFX (DIFFR) DIFFY (DIFFTHT) DIFFZ

*ECLIPSE 300 only*

- HEATTX
- HEATTY
- HEATTZ

*ECLIPSE 300 only*

- ROCKV
- BVRMX
- BVRMY

*ECLIPSE 300 only*

- BVRMZ
- BULKV
- DTPX1
PROPS section arrays

- SWL  SWCR  SWU
- SGL  SGCR  SGU
- KRW  KRO  KRG
- PCG  PCW

REGIONS section

- SATNUM  PVNUM  EQLNUM
- IMBNUM  FIPNUM  ENNUM
- ROCKNUM  MISCNUM  KRNUM
- WH2NUM  WH3NUM

ECLIPSE 300 only

- EOSNUM

SOLUTION section arrays

- PRESSURE  SWAT  SGAS
- RV  RS  TBLK

ECLIPSE 300 only

- SOIL  XMF  YMF

ECLIPSE 300 only

- TEMP  PBUB  PDEW

ECLIPSE 100 only

- GI

Notes

- Keywords in brackets are for radial geometry runs
- Operations on integer arrays can lead to unwanted results, since the operations are performed with floating (double) point precision and thereafter translated to an integer representation.
- Extreme caution should be exercised for operations on TRANX, TRANY and related keywords: X- and Y-transmissibilities reflect cell positions, and have zero values at the edges of the grid.
- See also keywords BOX, ENDBOX, EQUALS, ADD and MULTIPLY.
- If a MAXLIM operation is chosen, this will apply a maximum limit to an array in the current box. Refer also to keyword MAXVALUE.
- If a MINLIM operation is chosen, this will apply a minimum limit to an array in the current box. Refer also to keyword MINVALUE.
• It is possible to use this keyword on arrays between different sections of the dataset. This will enable the use of an operation such as \(SWL = Fn(PORO)\). For use of the \texttt{OPERATE} keyword in either the \texttt{PROPS} or \texttt{SOLUTION} section of ECLIPSE 100, as well as being able to use existing properties from these data sections, only certain grid properties can be “operated on”. For instance, \texttt{PORO, PORV, DEPTH, PERMX, PERMY} and \texttt{PERMZ}. Similar operations apply to \texttt{SOLUTION} section use inside ECLIPSE 300. In a one- or two-dimensional case, directional information for the collapsed dimensions is not available for use in the \texttt{SOLUTION} section (so, for instance, you cannot use \texttt{PERMX} in the \texttt{SOLUTION} section of a dataset which has only one cell in the x-direction).

• For \texttt{REGIONS} section use in both simulators, it is possible to “operate” on \texttt{MULTNUM, FLUXNUM} and \texttt{OPERNUM} regions (in order of preference).

• It is possible to specify a work array (see "Example 3") in place of the result array (item 1) and/or the argument array (item 9). Work arrays of double-precision or integer type may be specified, as required by the desired operation. Double precision work array names must begin with the characters ‘WORK’ followed by an integer specifying the index of the array that is the first work array should be named \texttt{WORK1}, the second \texttt{WORK2}, and so on. Integer work array names must begin with the characters ‘IWORK’ followed by an integer specifying the index of the array that is the first integer work array should be named \texttt{IWORK1}, the second \texttt{IWORK2}, and so on. The total number of double precision work arrays used must not exceed item 8 of the \texttt{REGDIMS} keyword, and the total number of integer work arrays used must not exceed item 9 of the \texttt{REGDIMS} keyword. Note that work array elements will be automatically initialized to zero (in ECLIPSE 100, work arrays are reset to zero at the start of each section of the datafile, or when changing between grids, for instance between two local grids or the global grid and a local grid).

• In the \texttt{SOLUTION} section, this keyword is processed at the very end after initialization has been done. It’s therefore important to be careful when combining its use with keywords such as \texttt{EQUALS, ADD} and \texttt{MULTIPLY}. These keywords will always be applied first, even if \texttt{OPERATE} appears before them in the dataset. \texttt{OPERATE} is processed before \texttt{OPERATER}, if both are present.

• Because this keyword is processed after initialization, array names for use in the \texttt{SOLUTION} section are output mnemonics rather than input keywords. For instance, to manipulate grid block temperatures you specify \texttt{TEMP} rather than \texttt{TEMPI}.

**Examples**

**Example 1**

Set \(DY\) equal to \(DX\):

```
OPERATE
  --result box  operate  param
  DY    6*   'COPY'    DX   /
  /
```

**Example 2**

Implement correlations over parts of grid:

```
----in box 1-11 1-19 1-4
OPERATE
  PERMX 1 11 1 19 1 4 'MULTA'  PORO   .444 .5555 /
  PERMX 1 11 1 19 1 4 'POLY'    PERMX .234 2 /
  PERMX 1 11 1 19 1 4 'POLY'    PERMX .034 3 /
  PERMX 1 11 1 19 1 4 'POLY'    PERMX .004 4 /
  PERMX 1 11 1 19 1 4 'SLOG'    PERMX .000 1.0 /
  PERMX 1 11 1 19 1 4 'MULTX'   PERMX 2.44 /
```
In the `REGIONS` section:

```plaintext
OPERATE
  SATNUM 6* COPY FIPNUM /
```

**Example 3**

Use of work arrays (item 8/9 of `REGDIMS` must be appropriately specified). For this example, since two double precision work arrays (WORK1, WORK2) are used, item 8 of `REGDIMS` should be set to 2; and since only 1 integer work array (IWORK1) is needed, then item 9 of `REGDIMS` should be set to 1:

```plaintext
---result box operate argument param
OPERATE
  WORK1 6* 'MULTX' PORO .45 /
  WORK2 6* 'LOG10' PERMX /
  WORK2 6* 'MULTIPLY' WORK1 /
  PERMY 6* 'ABS' WORK2 /

---result box operate argument param
OPERATE
  IWORK1 6* 'MULTX' SATNUM 2 /
  FIPNUM 6* 'ADDX' IWORK1 1 /
```
OPERATER

Performs operations on arrays for a specific region

This keyword is used to perform arithmetic operations on property arrays for a specific region.

The keyword may be followed by any number of records, each of which is terminated by a slash (/). The data is terminated by a blank record, containing only a slash.

Each record consists of:

1. The result array, \( R_i \)
2. The region number on which to perform the operation (by default this is from `OPERNUM` data, but item 7 below allows the use of other sets of region numbers).
   
   DEFAULT: 0
3. The type of operation. The table below describes valid operations. Let \( X_i \) denote the \( i \)th element of the argument array, and \( R_i \) the \( i \)th element of the result array. The Greek letters \( \alpha \) and \( \beta \) are scalar parameters. Examples for valid arrays are given later. All operations are performed with floating point arithmetic. If the result is an integer array, the real value is rounded to the nearest integer.

<table>
<thead>
<tr>
<th>Result</th>
<th>item 3</th>
<th>item 4</th>
<th>item 5</th>
<th>item 6</th>
</tr>
</thead>
<tbody>
<tr>
<td>( R_i = \alpha X_i + \beta )</td>
<td>‘MULTA’</td>
<td>( X )</td>
<td>( \alpha )</td>
<td>( \beta )</td>
</tr>
<tr>
<td>( R_i = R_i + \alpha X_i \beta )</td>
<td>‘POLY’</td>
<td>( X )</td>
<td>( \alpha )</td>
<td>( \beta )</td>
</tr>
<tr>
<td>( R_i = 10^{(\alpha + \beta X_i)} )</td>
<td>‘SLOG’</td>
<td>( X )</td>
<td>( \alpha )</td>
<td>( \beta )</td>
</tr>
<tr>
<td>( R_i = \log_{10}(X_i) )</td>
<td>‘LOG10’</td>
<td>( X )</td>
<td>n/a</td>
<td>n/a</td>
</tr>
<tr>
<td>( R_i = \ln(X_i) )</td>
<td>‘LOGE’</td>
<td>( X )</td>
<td>n/a</td>
<td>n/a</td>
</tr>
<tr>
<td>( R_i = \frac{1}{X_i} )</td>
<td>‘INV’</td>
<td>( X )</td>
<td>n/a</td>
<td>n/a</td>
</tr>
<tr>
<td>( R_i = \alpha X_i )</td>
<td>‘MULTX’</td>
<td>( X )</td>
<td>( \alpha )</td>
<td>n/a</td>
</tr>
<tr>
<td>( R_i = \alpha + X_i )</td>
<td>‘ADDX’</td>
<td>( X )</td>
<td>( \alpha )</td>
<td>n/a</td>
</tr>
<tr>
<td>( R_i = X_i )</td>
<td>‘COPY’</td>
<td>( X )</td>
<td>n/a</td>
<td>n/a</td>
</tr>
<tr>
<td>( R_i = \min(\alpha, X_i) )</td>
<td>‘MAXLIM’</td>
<td>( X )</td>
<td>( \alpha )</td>
<td>n/a</td>
</tr>
<tr>
<td>( R_i = \max(\alpha, X_i) )</td>
<td>‘MINLIM’</td>
<td>( X )</td>
<td>( \alpha )</td>
<td>n/a</td>
</tr>
<tr>
<td>( R_i = \alpha X_i \beta )</td>
<td>‘MULTP’</td>
<td>( X )</td>
<td>( \alpha )</td>
<td>( \beta )</td>
</tr>
<tr>
<td>( R_i = \text{abs}(X_i) )</td>
<td>‘ABS’</td>
<td>( X )</td>
<td>n/a</td>
<td>n/a</td>
</tr>
<tr>
<td>( R_i = X_i R_i )</td>
<td>‘MULTIPLY’</td>
<td>( X )</td>
<td>n/a</td>
<td>n/a</td>
</tr>
</tbody>
</table>

Table 3.22: Valid operation types

4. Array parameter depending on item 3. Must be supplied, no default.
5. Scalar parameter depending on item 3. Must be supplied if required, no default.
6. Scalar parameter depending on item 3. Must be supplied if required, no default.

7. The name of a fluid-in-place region set to be used instead of the OPERNUM region. This can be either an ordinary region defined in the REGIONS section, or a dynamic region defined in the SOLUTION section. This parameter can be used no earlier than the SOLUTION section, and, for dynamic regions, only after the region in question has been defined. The default is to use the OPERNUM region.

ECLIPSE 300

This keyword may be used in the SCHEDULE section. However, this is a specialist facility.

Note: In the SCHEDULE section, regions (pre-defined by the FIPxxxx keyword in the REGIONS section) can be used in the new item 7. This region will be used in place of the OPERNUM region.

Legal arrays

GRID section arrays

<table>
<thead>
<tr>
<th>DX (DR)</th>
<th>DY (DTHETA)</th>
<th>DZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>PERMX (PERMR)</td>
<td>PERMY (PERMHT)</td>
<td>PERMZ</td>
</tr>
<tr>
<td>MULTX (MULTR)</td>
<td>MULTY (MULTHT)</td>
<td>MULTZ</td>
</tr>
<tr>
<td>PORO</td>
<td>NTG</td>
<td>DZNET</td>
</tr>
<tr>
<td>TOPS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>DIFFMX (DIFFMR)</td>
<td>DIFFMY (DIFFMHT)</td>
<td>DIFFMZ</td>
</tr>
</tbody>
</table>

ECLIPSE 300 only

| MIDS |

EDIT section

<table>
<thead>
<tr>
<th>PORV</th>
<th>DEPTH</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRANX (TRANR)</td>
<td>TRANY (TRANTHT)</td>
</tr>
<tr>
<td>DIFFX (DIFFR)</td>
<td>DIFFY (DIFFHTHT)</td>
</tr>
</tbody>
</table>

ECLIPSE 300 only

<table>
<thead>
<tr>
<th>HEATTX</th>
<th>HEATTY</th>
<th>HEATTZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>ROCKV</td>
<td>BVRMX</td>
<td>BVRMY</td>
</tr>
<tr>
<td>BVRM2</td>
<td>BULKV</td>
<td>DTPX1</td>
</tr>
<tr>
<td>DTPX2</td>
<td>DTPY1</td>
<td>DTPY2</td>
</tr>
<tr>
<td>DTPZ1</td>
<td>DTPZ2</td>
<td></td>
</tr>
</tbody>
</table>

PROPS section

<table>
<thead>
<tr>
<th>SWL</th>
<th>SWCR</th>
<th>SWU</th>
</tr>
</thead>
<tbody>
<tr>
<td>SGL</td>
<td>SGCR</td>
<td>SGU</td>
</tr>
</tbody>
</table>
REGIONS section arrays

SATNUM  PVTNUM  EQLNUM
IMBNUM  FIPNUM  ENDNUM
ROCKNUM  MISCNUM  KRNUM
WH2NUM  WH3NUM

ECLIPSE 300 only

EOSNUM

SOLUTION section arrays

PRESSURE  SWAT  SGAS
RV  RS  TBLK

ECLIPSE 300 only

SOIL  XMF  YMF

ECLIPSE 300 only

TEMP  PBUB  PDEW

ECLIPSE 100 only

GI

SCHEDULE section arrays (specialist facility)

ECLIPSE 300 only

PORO  PERMX  PERMY

ECLIPSE 300 only

PERMZ  MULTX  MULTIY

ECLIPSE 300 only

MULTZ  SIGMA(V)  MULTSIG(V)

ECLIPSE 300 only

MULSGGD(V)  DIFFMMF  WORK(1/2)

Notes

• The OPERNUM array must be specified before the OPERATER keyword and its associated data.

• Keywords in brackets are for radial geometry runs

• Operations on integer arrays can lead to unwanted results, since the operations are performed with floating (double) point precision and thereafter translated to an integer representation.

• Extreme caution should be exercised for operations on TRANX, TRANY and related keywords: X- and Y-transmissibilities reflect cell positions, and have zero values at the edges of the grid.

• See also keywords EQUALS, ADD, COPY and MULTIPLY.

• If a MAXLIM operation is chosen, this will apply a maximum limit to an array in the current box. Refer also to keyword MAXVALUE.
• If a MINLIM operation is chosen, this will apply a minimum limit to an array in the current box. Refer also to keyword MINVALUE.

• It is possible to use this keyword on arrays between different sections of the dataset. This will enable the use of an operation such as SWL = Fn(PORO). For use of the OPERATER keyword in either the PROPS or SOLUTION section of ECLIPSE 100, as well as being able to use existing properties from these data sections, only certain grid properties can be “operated on”. For instance, PORO, PORV, DEPTH, PERMX, PERMY and PERMZ. Similar operations apply to SOLUTION section use inside ECLIPSE 300. In a one- or two-dimensional case, directional information for the collapsed dimensions is not available for use in the SOLUTION section (so, for instance, you cannot use PERMX in the SOLUTION section of a dataset which has only one cell in the x-direction).

• For REGIONS section use in both simulators, it is possible to “operate” on MULTNUM, FLUXNUM and OPERNUM regions (in order of preference).

• It is possible to specify a work array (see “Example 4”) in place of the result array (item 1) and/or the argument array (item 4). Work arrays of double-precision or integer type may be specified, as required by the desired operation. Double precision work array names must begin with the characters ‘WORK’ followed by an integer specifying the index of the array that is the first work array should be named WORK1, the second WORK2, and so on. Integer work array names must begin with the characters ‘IWORK’ followed by an integer specifying the index of the array that is the first integer work array should be named IWORK1, the second IWORK2, and so on. The total number of double precision work arrays used must not exceed item 8 of the REGDIMS keyword, and the total number of integer work arrays used must not exceed item 9 of the REGDIMS keyword. Note that work array elements will be automatically initialized to zero (in ECLIPSE 100, work arrays are reset to zero at the start of each section of the datafile, or when changing between grids, for instance between two local grids or the global grid and a local grid).

• To use this keyword with local grid array values, it must be specified inside an appropriate CARFIN or REFINE keyword. Note that this is only sensible for ECLIPSE 300, since for ECLIPSE 100 it is not possible to specify MULTNUM, FLUXNUM or OPERNUM data for a local grid.

• In the SOLUTION section, this keyword is processed at the very end stage, after initialization has been completed. It is therefore important to be careful when combining its use with keywords such as EQUALS, ADD and MULTIPLY. These keywords will always be applied first, even if OPERATER appears before them in the dataset. OPERATE is processed before OPERATER, if both are present.

• Because this keyword is processed after initialization, array names for use in the SOLUTION section are output mnemonics rather than input keywords. For instance, to manipulate grid block temperatures you specify TEMP rather than TEMPI.

### Additional notes

_ECLIPSE 300 only_

Additional notes for the use of this keyword in the SCHEDULE section. Again, this is a specialist facility. In the SCHEDULE section, the effect of this keyword will be as follows:

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DIFFMMF</td>
<td>Modifies the matrix-fracture diffusivity in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as DIFFMMFS.</td>
</tr>
<tr>
<td>MULSGGD</td>
<td>Modifies the matrix-fracture coupling transmissibility for oil-gas gravity drainage in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as MULSGGDS.</td>
</tr>
</tbody>
</table>
Keyword | Description
---|---
MULTSIG | Modifies the matrix-fracture coupling transmissibility in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as MULTSIGS.
MULTSIGV | 
MULTXZ | Modifies transmissibility in the X/Y/Z-direction. If output is required for flexible restart in the model, the multiplier values will be output as MULTXS/MULTYS/MULTZS.
MULTY | 
MULTZ | 
PERMX | Modifies permeability in the X/Y/Z-direction. Also approximates the corresponding transmissibility multiplier MULTX/Y/Z determined from the ratio of the new permeability value to the old one. If output is required for flexible restart in the model, the permeability values will be output as MODPERMX/Y/Z.
PERMY | 
PERMZ | 
PORO | Modifies porosity. Modifies the corresponding pore volume by a multiplier MULTPV determined from the ratio of the new porosity value to the old one. If output is required for flexible restart in the model, the porosity values will be output as MODPORO, and the pore volume multipliers will be output as MULTPVS.
SIGMA | Modifies the matrix-fracture coupling transmissibility in dual-porosity runs. If output is required for flexible restart in the model, these multiplier values will be output as SIGMA.
SIGMAV | 
WORK (1/2) | These are work arrays to be stored for subsequent use. If output is required for flexible restart in the model, these values will be output as WORK (1/2).

Table 3.23: Additional notes for OPERATER in the SCHEDULE section

ECLIPSE 300 only

Output to the flexible restart file, of the quantities DIFFMMFS, MULSSGDS, MULTSIGS, SIGMA and WORK (1/2), is not currently supported for LGRs.

Examples

Example 1

Set DY equal to DX:

```
OPERATER --result region operate param
DY 2 'COPY' DX /
```

Example 2

Implement correlations over different regions of the grid:

```
OPERATER
PERMX 1 'MULTA' PORO .444 .5555 /
PERMX 2 'POLY' PERMX .234 2 /
PERMX 2 'POLY' PERMX .034 3 /
PERMX 2 'POLY' PERMX .004 4 /
PERMX 1 'SLOG' PERMX .000 1.0 /
PERMX 1 'MULTX' PERMX 2.44 /
PERMZ 3 'COPY' PERMX /
```

Keywords

1518
Example 3

In the REGIONS section:

```
OPERATER
SATNUM 2 COPY FIPNUM /
/
```

Example 4

Use of work arrays (item 8 of REGDIMS must be appropriately specified).

```
--result reg operate arg param
OPERATER
WORK1 1 'MULTX' PORO .45 /
WORK2 1 'LOG10' PERMX /
WORK2 1 'MULTIPLY' WORK1 /
PERMY 1 'ABS' WORK2 /
/
--result box operate argument param
OPERATER
IWORK1 1 'MULTX' SATNUM 2 /
FIPNUM 1 'ADDX' IWORK1 1 /
/
```

For the example above, since two double precision work arrays (WORK1, WORK2) are used, item 8 of REGDIMS should be set to 2; and since only 1 integer work array (IWORK1) is needed, then item 9 of REGDIMS should be set to 1.
**OPERNUM**  
**Defines regions for performing operations on arrays**

The **OPERNUM** keyword provides a region definition for performing arithmetic operations on property arrays using the **OPERATER** keyword.

The **OPERNUM** array must be specified before the **OPERATER** keyword and its associated data.

To use the **OPERNUM** keyword, the maximum number of **OPERNUM** regions must first be defined. To do this, item 7 in the **REGDIMS** keyword in the **RUNSPEC** section must be set to a value greater than zero.

The **OPERNUM** keyword should be followed by one integer for each grid cell, specifying the region number to which it belongs. The data must be terminated by a slash (/).

Any cells not explicitly assigned a value of **OPERNUM** will be taken to be in region 0, which implies that no operation is allowed for this particular cell.

The **OPERNUM** array can be copied into the **REGIONS** section arrays (**FIPNUM** and **PVNUM**) using the **COPY** keyword. However, it cannot be set in this way - it can only be used as the “from” array.

The **OPERNUM** keyword and its data should be present in the **GRID** section in order to use the **OPERATER** keyword in the **GRID**, **EDIT** or **PROPS** section. The **OPERNUM** keyword can be re-specified in the **REGIONS** section for use in the **REGIONS** and **SOLUTION** sections.

The **OPERNUM** array can be used with keywords **MULTIREG**, **ADDREG**, **EQUALREG** and **COPYREG**.

---

**Note:** The **OPERNUM** keyword cannot be used inside a CARFIN/RADFIN...ENDFIN bracket, so it is currently not possible to use **OPERNUM**, together with **OPERATER**, to work on specific LGRs. To do this use the **OPERATE** keyword instead.

---

For use with the Dual Porosity option, only the matrix cells or the fracture cells need to be input. The cell values in the other porosity will be copied automatically. If both matrix and fracture cells are given, a warning message will be output complaining about too much data. If the user wants to specify matrix cells and fracture cells separately, then item 117 in **OPTIONS** should be set greater than 1. This will also affect use of the **MULTNUM**, **PINCHNUM** or **FLUXNUM** keywords.

When **OPERNUM** is defined, both matrix and fracture cells should be provided for use with the dual porosity option if operations are required on fracture cells. There is no copying of **OPERNUM** data from the matrix to fracture cells.

---

**Example**

To define 9 separate regions in a 4*4*3 field (with the maximum number of **OPERNUM** regions set equal to 9):

```
OPERNUM
1 1 2 2  
1 1 2 2  
3 3 3 3  
4 4 4 4  
5 5 5 5  
6 7 8 8  
6 7 8 8  
6 7 8 8  
16*9 / 
```

---

**Keywords**

**ECLIPSE 100**

**ECLIPSE 300**

---

1520
**OPTCONS**

Specify optimization constraints

This keyword is used to specify the constraints to be observed during an optimization run.

The keyword must be followed by a set of records each specifying a constraint to be observed in the optimization. If the keyword is not present then the optimization is unconstrained.

This keyword should only appear once in the data file.

Each record should be terminated with a slash (/) and the set of records must end with a slash.

1. Mnemonic specifying quantity to be constrained. These mnemonics are drawn from the summary vector mnemonics. Available mnemonics are.

   - **FOPR**  Field oil production rate
   - **FWPR**  Field water production rate
   - **FWIR**  Field water injection rate
   - **FGPR**  Field gas production rate
   - **FGIR**  Field gas injection rate
   - **FLPR**  Field liquid production rate
   - **FGOR**  Field gas oil ratio
   - **FWCT**  Field water cut
   - **GOPR**  Group oil production rate
   - **GWPR**  Group water production rate
   - **GWIR**  Group water injection rate
   - **GGPR**  Group gas production rate
   - **GGIR**  Group gas injection rate
   - **GLPR**  Group liquid production rate
   - **GGOR**  Group gas oil ratio
   - **GWCT**  Group water cut
   - **WOPR**  Well oil production rate
   - **WWPR**  Well water production rate
   - **WWIR**  Well water injection rate
   - **WGPR**  Well gas production rate
   - **WGIR**  Well gas injection rate
WLPR  Well liquid production rate
WGOR  Well gas oil ratio
WWCT  Well water cut

2. Unique identifier for the domain to which the constraint specified in the first item applies. This should
be either just ‘FIELD’, or the name of a well or group which has already been defined in the
SCHEDULE section. The wildcard character ‘*’ may not be used to specify more than one well or
group at the same time. Well lists, specified using the WLIST keyword, cannot be used as the domain
identifier.

3. The type of inequality constraint. This may be set to either ‘<’ or ‘>’ (or ‘LE’ or ‘GE’), depending on
whether the constraining value is an upper or a lower bound to the quantity.
DEFAULT: ‘<’ or ‘LE’

4. The value of the constraint, which is the limiting value of the mnemonic specified using items 1 and 2
of the keyword.

5. Index of first report time for this constraint.
DEFAULT: 0 (or the first report index in a RESTART case)

6. Index of last report time for this constraint.
DEFAULT: The last report index requested in the SCHEDULE section.

Example

```
-- Constrain gas production from well P1,
-- and water injection to well INJE2
'WGPR' P1  '<' 2000.0 /
'WWIR' INJE1  '>' 500.0 /
```
Specify optimization dimensions

This keyword is used to specify user-defined limits on the size of the optimization problem being considered.

The keyword is followed by a single record consisting of two integers. The record is terminated with a slash (/):

1. The maximum number of outer optimization iterations permitted during the optimization. (Each of these iterations includes calculation of gradients by the adjoint method.)
   DEFAULT: 0

2. The maximum number of simulations that may be performed in total during the optimization.
   DEFAULT: 1

**Note:** Due to these defaults, if the OPTDIMS keyword does not appear in a dataset, then the optimization gradients are calculated, and the run terminated normally.

### Example

```
OPTDIMS -- MOPITS MOPSIM
  10   20   /
```
**OPTFUNC**

**Specify objective function for optimization**

This keyword is used to specify the objective function to be optimized in an optimization run of the simulator. This objective function may be composed of several separate component functions, giving flexibility to the engineer.

The keyword must be followed by a set of records each specifying a separate linear component of the objective function. Each record should be terminated with a slash (/) and the set of records must end with a slash. At least one record is required.

This keyword should only appear once in the data file.

1. Mnemonic specifying quantity to be included in the objective function. Implemented mnemonics are:
   - FOPT  Field oil production total
   - FWPT  Field water production total
   - FWIT  Field water injection total
   - FGPT  Field gas production total
   - FGIT  Field gas injection total
   - FLPT  Field liquid production total
   - GOPT  Group oil production total
   - GWPT  Group water production total
   - GWIT  Group water injection total
   - GGPT  Group gas production total
   - GGIT  Group gas injection total
   - GLPT  Group liquid production total
   - WOPT  Well oil production total
   - WWPT  Well water production total
   - WWIT  Well water injection total
   - WGPT  Well gas production total
   - WGIT  Well gas injection total
   - WLPT  Well liquid production total

2. Unique identifier for the domain to which the mnemonic specified in the first item applies.
   - For group and well quantities, this should be the name of a group or well. These names should already have been defined in the SCHEDULE section.
   - For field type objective function components, this entry assumes the value of ‘FIELD’.

---

**Keywords**

1524
It is **not** possible to use the wildcard character '*’ to specify more than one well or group. Well lists, specified using the **WLIST** keyword, cannot be used as the domain identifier.

3. Coefficient specifying the weight of this component function in the objective function. Negative values are allowed.
   Default: 1.0

4. Daily discount rate (as a percentage) to be used in calculating the discount factor for this component.
   Default: 0.0

5. Index of first report time at which this component is part of the objective function.
   Default: 0 (or the first report index in a **RESTART** case)

6. Index of last report time at which this component is part of the objective function.
   Default: The last report index requested in the **SCHEDULE** section.

**Example**

Objective function is to optimize field oil production while penalizing water production from well 'PROD' between report time indices 2 and 7:

```
OPTFUNC
'FOPT' FIELD /
'WWPT' PROD -1.0 0.0 2 7/
/
```

**Keywords**

1525
## OPTIONS

Activates special program options

The **OPTIONS** keyword can be used to activate special options within particular facilities in ECLIPSE 100. These options are mainly of a temporary or experimental nature, or act to restore back-compatibility with earlier versions of the code.

The keyword should be followed by a number of integers, each of which activates a special option within particular ECLIPSE 100 facilities. A value equal to zero switches off the special option. A value other than zero activates a special option. The data field should be terminated by a slash (/). If fewer than the maximum number integers are read, the remainder are left unchanged. Once an option is activated, it remains activated and retains its value in all subsequent uses of **OPTIONS** keyword unless it is explicitly changed or turned off. Note that repeat counts (for example 3*0) can be used if required (there must be no spaces before or after the asterisk).

### Control

1. If >0 prevents RS being reset to RSSAT of grid block during the equilibration calculation when the grid cell contains free gas.

2. **Well predictor switch**
   
   (<0 off, =0 use as needed, =n use for NEWTIT ≤ n).

3. **Not used at present.**

   The default setting of principal directions for the linear solver can be overridden using the **SOLVDIRS** keyword.

4. If =1 the critical rather than the connate water saturation is used in the simulation part of a VE run to calculate the contact depths. This prevents very small saturations being mobile through the base of the cell, but the VE Equilibration option is no longer static.

   If =2 the critical rather than the connate saturation is used, as above, however it is only applied to the calculation of the water contact depths. This prevents unrealistic residual oil saturations in the gas zone of three-phase runs.

5. If =1 the critical rather than the connate gas saturation is used in the simulation part of a VE run to calculate the contact depths. This prevents very small gas saturations being mobile through the top of the cell, but the VE equilibration option is no longer static. This may prevent convergence difficulties when bubble point crossing occurs in a VE run.

   If =2 the critical rather than the connate saturation is used, as above, however it is only applied to the calculation of the gas contact depths. This prevents unrealistic residual oil saturations in the water zone of three-phase runs.

6. **Not used at present.**

   It is possible to turn off capillary pressure hysteresis using the 4th data item in the **EHYSTR** keyword.

7. **If <0 Saturation chops are suppressed.**

   If >0 Oil saturation chops for **VAPPARS** are also suppressed.

8. **Chop on pressure change in well solution iterations to limit drawdown reversal (<0 off, ≥ 0 on for segmented head option).**

9. Implicit/explicit water and gas fraction in THP calculations

   ≤0 implicit, >0 explicit. See keyword **WVFEXP**.

### Keywords

<table>
<thead>
<tr>
<th>x</th>
<th>ECLIPSE 100</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>ECLIPSE 300</td>
</tr>
<tr>
<td></td>
<td>SPECIAL</td>
</tr>
<tr>
<td>x</td>
<td>RUNSPEC</td>
</tr>
<tr>
<td></td>
<td>GRID</td>
</tr>
<tr>
<td></td>
<td>EDIT</td>
</tr>
<tr>
<td></td>
<td>PROPS</td>
</tr>
<tr>
<td></td>
<td>REGIONS</td>
</tr>
<tr>
<td></td>
<td>SOLUTION</td>
</tr>
<tr>
<td></td>
<td>SUMMARY</td>
</tr>
<tr>
<td>x</td>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>
If <0 Causes immediate shut-in if flow is negative in any well iteration.
10. If >0 chops on P, R, and Rv are suppressed.
11. If >0 produces quiescent initial conditions for gravity drainage/imbibition calculation in dual porosity runs.
12. Not used at present.
   The transition zone end-point scaling is now controlled by the TZONE keyword in the PROPS section.
13. Not used at present.
   The alternative relative permeability scaling option, which preserves the value of $K_{rw} (S_w = 1 - S_{or})$ etc. can be initiated by using the SCALECRS keyword on the PROPS section.
14. Flag for controlling formatted data compression output to Restart files. The formatted output keyword FMTOUT in the RUNSPEC section must be set to true for this option to work.
   If = 0 sets compression off for formatted output.
   If = 1 sets normal compression on for formatted output, that is only puts a repeat count in front of repeated numbers, but unwanted zeros are not scanned and taken out before the numbers are written out to file.
   If = 2 sets full compression on for formatted output, that is the same as flag = 1, but scans for unwanted zeros and take them out before the numbers are written out to file.

   Setting the flag to 2 increases the CPU time. Therefore only set to full compression if saving disk space is more important than an increase in CPU time.
15. Number of iterations within a timestep in which a well can be revived after being closed because it cannot operate at the reservoir conditions of the previous iteration (for example under THP control). If = 0 defaults to the usual number of 3 iterations. If < 0 prevents the well being revived.
16. Not used at present.
17. Not used at present.
   Pinchout transmissibility calculations are now controlled by the PINCH keyword.
18. Not used at present.
   The use of the VE pseudo-capillary pressures can now be controlled with the VEFRACP or VEFRACPV keyword.
19. If set =1, with the vertical equilibrium option switched on, then the original method of calculating interblock flows taking no account of vertical displacements (for example at faults) will be used (see “Vertical equilibrium” in the ECLIPSE Technical Description). Note that this option can have large effects in non-faulted cases specified using block centered input.
   If set =2, with a block centered grid and the Vertical Equilibrium option switched on, then the flows are calculated assuming a corner point approximation to the original grid. This option is used in local refinement cases in which the global grid is specified as a block centered grid. (See “Vertical equilibrium” in the ECLIPSE Technical Description)
   DEFAULT:
   0 In corner point geometry cases,
1 In block centered geometry cases.

20. If set >0, the well and group water phase guide rates are increased if necessary to ensure that \( GR_w / GR_o > X \), where X is the reciprocal of the number entered for this control. This increases production from the wells with lower WORs if a group or the field changes from oil rate control to water rate control.

**CAUTION:** If the production distribution is changed too much (by making the minimum water-oil guide rate ratio too large), the group or field may not be able to obey both its oil and water production limits.

21. If set >0, Liquid rate controls refers to Oil + Gas instead of Oil + Water. The redefinition of liquid rate to mean total hydrocarbon surface rate applies to:

1 Well liquid rate limits set in **WCONPROD** and **WELTARG**.

2 Group liquid rate limits set in **GCONPROD** and **GCONPRI** unless the action on violation is well closure or workover

3 Liquid phase guide rates set in **WGRUPCON** and **GCONPROD**

4 The control phase LIQ selected in item 7 of **WCUTBACK**.

This option cannot be used with **FIELD** units, because the oil and gas rates are measured in different quantities.

22. Maximum number of times a hunting warning is printed for a given well. If it hunts any more it is shut or stopped, according to item 9 of keyword **WELSPECS**. A value of 0 or negative implies no limit.

23. Sets the maximum number of times certain messages are printed. This option controls some warnings about the grid. A value of 0 sets the maximum to the default value of 15. A negative value implies no limit.

24. Not used at present

The pore volume and transmissibility multipliers for the rock compaction option can be tabulated against the effective overburden pressure by using the item 1 in the **ROCKOPTS** keyword.

25. If set >0, drainage regions are written to the **DEBUG** file at each report time.

These are useful for the ECLIPSE 200 Flux Boundary option. Flows are minimal between different drainage regions. Thus if the drainage regions are reasonably stable over a period of time, they can be used with the flux boundary option to save computing time in sensitivity or history matching runs.

If set >1, the location of the sinks (typically wells) within each drainage region is identified by a negative region number.

The computer time used by this option is reported if the argument **DRND** in **RPTSCHED** has been set.

26. If set >0, a more relaxed criterion for switching solution variables is used in cases when a water cell has some hydrocarbon influx. This can be helpful in a few rare cases when negative gas saturations can develop near a water/HC boundary.

27. If set >0, the run is stopped if a timestep fails to converge and the timestep is too small to be chopped (see **TUNING** keyword).
If set >9 then the run is stopped following any timestep convergence failure.

28. If set >0 with capillary pressure hysteresis option, then saturation chops will be used for cell that change hysteresis state in the gas phase.

If set >1 then chops are applied to the water phase as well.

29. Not used at present.

30. If set >0, a binary chop is implemented in the well iterations, restricting the solution increments if the residuals get worse. This could help the convergence of the well iterations in certain difficult cases. However, for multisegment well convergence problems you are recommended to try the WSEGITER keyword before changing this switch.

If set < 0 binary chops are not applied.

If set = 0 (the default) binary chops are applied in multisegment wells but not in standard wells.

If set >0 binary chops are applied in all types of well.

If set >1 an additional pressure chop are applied in multisegment wells, at the value which would cause a drawdown reversal. This is only applied when the connection depth is the same as the depth of the segment’s node.

31. If set >0, the average field and region pressure output is the pore-volume weighted-average, instead of the hydrocarbon pore-volume weighted-average. The pore volumes used are those at reference conditions (that is uncompacted pore volumes).

The default average field pressure is

\[ FPR = \frac{\sum HCPV \cdot p}{\sum HCPV} \]

where \( HCPV = PV(1-Sw) \) is the hydrocarbon pore volume and \( p \) is the oil phase pressure. Setting OPTIONS (31)>0 implies

\[ FPR = \frac{\sum PV \cdot p}{\sum PV} \]

Note that the definition of summary keywords \( FPRH, RPRH, FPRP, RPRP \) is independent of this switch.

This switch has no effect on the reported average \( Rs \) and \( Rv \) that are defined, for example, by

\[ FRS = \frac{\sum PV \cdot Rs \cdot So}{\sum PV \cdot So} \]

In addition, if the absolute value of the switch is divisible by 2 and > 0, then the field and average potentials output are the pore-volume weighted-averages, rather than phase pore-volume weighted-averages (which is the default).

As detailed in "Potential calculations" in the ECLIPSE Technical Description, the default potentials reported are (for oil)

\[ FPPO = \frac{\sum \Phi_o \cdot S_o \cdot PV}{\sum S_o \cdot PV} \]

where \( \Phi_o \) is the oil potential and \( S_o \) is the oil saturation.
If \([OPTIONS(31)]\) is divisible by 2 and \(OPTIONS(31) > 0\), then the field oil potential is given by

\[
FPPO = \frac{\sum \Phi_o \cdot PV}{\sum PV}
\]

32. Number of times a single timestep can be consecutively chopped because a well has switched from rate control to pressure control and died in the same timestep.

If \(\text{If} = 0\) defaults to 1.

If \(\text{If} = \text{negative}\) prevents timestep being chopped for this reason. In LGR runs, the global timestep is not chopped for wells in local grids - only the local timesteps are chopped. Setting the control value to between 11 and 20 allows chops of the global timestep, the maximum number of times being equal to the control value minus 10.

33. Number of network balancing iterations in which a well can be revived after being closed because it cannot operate against its current THP.

If \(\text{If} = 0\), this defaults to max \((4, \text{NETBAL}/2)\) where \(\text{NETBAL}\) is the maximum number of network balancing iterations allowed (see the \text{NETBALAN} keyword in the Network option).

34. If \(\text{If} > 0\) when running the Gas Field Operations Model, prevents wells being opened automatically from the drilling queue during the second pass of the contract period, in the same month that they were opened in the 1\(^{st}\) pass, if they are not actually needed to meet the DCQ x profile target.

35. If \(\text{If} > 0\) ECLIPSE will revert to the pre-98A method of calculating the explicit wellbore head (\text{SEG in WELSPECS Item 12}) when API tracking is used, that is, the head is calculated from the average oil surface density in the well instead of an upstream flow weighted average at each connection.

36. If \(\text{If} > 0\) with the nine-point scheme active, matrix preconditioning is suppressed (for development testing).

37. If \(\text{If} > 0\) ECLIPSE replaces the solution state of each local grid cell with that of the global host cell when updating the global solution. This can be useful in cases where a change of state in small local cells takes a longer timescale to be mirrored in the global host cell.

One example is a gas condensate case where a radial refinement is used around the well. The small cells nearest the well connections may change from gas only to gas and liquid at the start of the simulation, whereas the larger cells do not make this change until later in the simulation.

Using this option allows ECLIPSE to weight these changes of state by the volume of the cells, essentially smoothing over these transient features.

38. If \(\text{If} > 0\) the oil gravity terms are calculated using the region average oil reservoir density, rather than the local reservoir density. This can prevent numerically induced flows that may cause convergence problems in some cases.

\textit{Note:} The use of this switch is not recommended in general.

39. If \(\text{If} > 0\) ECLIPSE allows regions of different constant \(R_s\) or \(R_v\) to be in contact. This overrides the default behavior that only permits contact between regions of equal constant \(R_s\) or \(R_v\). Any such connections are now reported as warnings instead of errors.

40. If \(\text{If} > 0\) the well potential rates are calculated ignoring any THP limits. This option may be useful if the potentials are not used for anything important and the potential flows would extrapolate the VFP tables.
41. If set >0 the well inflow calculation is modified in Vertical Equilibrium runs. The connection depth will be moved to the center of the perforated range within the grid block, and the grid block pressure is corrected to this depth using the hydrostatic gradient of the phases.

42. Time average weighting factor (NWT) for $B_o$ and $R_s$ in flash transformation facility (see keyword COMPFLSH):

$$B_o \text{ (used)} = \frac{(B_o \text{ cell} + |NWT| \cdot (B_o \text{ (previous)}))}{|NWT|}$$

Set to 2 or greater to damp out any oscillations. A negative value (for example -1) restores the pre-96A treatment of undersaturated oil. Note that a value < -1 also activates time-averaging of the transformation terms.

43. Not used at present.

The rock data (ROCK keyword) corresponds to the saturation table (SATNUM) region numbers rather than the PVT region numbers (PVTNUM) if item 3 of the ROCKOPTS keyword is set to the string SATNUM.

44. Controls the treatment of the oil density at the water oil contact. If the control is set < 0 then the oil density for flow from a cell containing oil to a 100% water cell is set equal to the average oil density in the two cells. In the water cell the oil is assumed to be saturated. The default case (control = 0) uses the oil density in the oil cell only. The control only has effect on runs containing oil with dissolved gas. Versions of ECLIPSE prior to release 95A used the former method.

45. If set >0, the Kh or denominator term is not calculated from the connection factor if a positive value for this is entered in item 8 of keyword COMPDAT. This option can be used to achieve back-compatibility with pre-95A datasets, if required.

46. Controls the interpolation of $MINPV$ or $MINPVV$ to LGRs when either $MINPV$ or $MINPVV$ is specified in the global grid but not in local grids. If set to 1, the threshold pore volumes are not copied from global host cells, which is the default for 95A. This reproduces the behavior of pre-95A datasets where the threshold pore volume is defaulted to $10^{-6}$ in local grids for which it is not explicitly defined.

47. If set > 0, prevents well connection pressures from falling below atmospheric pressure. This may be useful for low pressure pumped wells in which there is an oil-gas interface within the perforated section.

48. If set > 0 in a dual porosity single permeability run, then if any fracture cells are made inactive because the fracture cell pore volumes fall below the threshold set by $MINPV$ or $MINPVV$, the corresponding matrix cells are also inactivated.

If set >1, then any active matrix cells with a corresponding inactive fracture cell are made inactive, for whatever reason the fracture cell is inactive (whether or not this is due to $MINPV$ or $MINPVV$).

49. If set > 0, then well efficiency factors are not taken into account when summing the well ALQs to compare the group lift gas rates with their maximum lift capacities set in keyword GLIFTLIM. This restores back compatibility with pre-96A versions of ECLIPSE, in which efficiency factors are not taken into account.

50. If set > 0, then the nine-point scheme is only applied in the areal dimensions. The option is not used at present.

51. If set = 1, temporal interpolation is used for the first local solution following a global timestep, and the state variable is reset in the global halo cells surrounding the LGR during temporal interpolation. This may improve convergence if the LGR takes several mini-steps to synchronize with the global timestep.
If set = 2, the initial local solution is reset to the values at the end of the previous local timestep when solving the local grids. This was the method used in versions previous to 95A. If set = 3, in addition the flows are recomputed for the first iteration on the local grid.

52. If set = 1, then the input corner point depths for a \textit{VE} run will always be honored as for versions previous to 95A.

If set = 2, the bottom corner point depths for each cell are modified to be consistent with the top depths of the next active cell below in the same column. The number of warning messages issued where this occurs may be limited using \texttt{OPTIONS} switch 23.

If set = 0 (the default), the depths are modified to ensure there is no vertical gap between adjacent local and global grid cells on the top and bottom faces of local grids. This is required to calculate the correct solution in local grid cases, for example when a pinched-out layer of non-zero thickness lies adjacent to the top or bottom faces of a local grid.

53. If set > 0, the closure of additional well connections “below” the offending connection in +CON workovers for standard wells is governed purely by their true depth values. This restores the pre-2002A treatment of these workovers.

If set = 0 (the default), the additional connections to be closed are selected according to the connection ordering as discussed in keyword \texttt{COMPORD}.

Note that friction wells and multisegment wells always close their connections by their position order along the wellbore, regardless of the setting of this switch.

54. Not used at present.

The control of whether or not well efficiency factors are taken into account in the network branch flows in the extended network model is now governed by item 3 of the \texttt{WEFAC} keyword.

55. In the wellbore friction option this can be used to damp the Newton iteration of the well solution if well convergence problems arise. In certain cases, incrementing the well solution by only part of the Newton step, possibly in addition to increasing \texttt{MXWSIT} in the \texttt{TUNING} keyword, may overcome convergence problems.

If set = n then the increment predicted by the Newton method is damped by a factor of $1/(1+n)$. This factor is only applied to the friction wells and values other than $n=1$ should only be needed in extreme circumstances.

Since this option is primarily intended to overcome convergence problems that sometimes arise when friction wells are initially opened, it is \textit{very} inefficient to leave this option set once the well in question has been successfully opened as, after this time it does not aid, but hinder, the convergence of the well equations.

56. If set >0, the action of any or \texttt{LGRON} keywords in Local Grid Refinement runs is performed before the global grid is solved. By default, the action is applied after the global grid is solved but before the local grids are solved.

If set > 1, the solver weighting factors are set equal to those of the unrefined grid whenever \texttt{LGROFF} is encountered.

This switch is ignored for in-place local grids, in which the action is always applied before solving the coupled system.

57. This switch invokes an alternative procedure for handling automatic chokes in the Extended Network option when a low flow target has been set, which may be tried if there are difficulties with the default method. It is quite possible for the choke’s pressure drop iterations to change the pressure at the wellheads sufficiently that all the wells die together. To avoid this happening ECLIPSE ‘chops’ the...
iteration in the range of pressures it calculates the wells stop flowing at. By default the well’s ‘zero flow’ THP is calculated by assuming all well variables are fixed, including the BHP, and then calculating the THP at which a zero flow is obtained.

If set = 1 then ECLIPSE calculates the actual THP at which the well would die, now only assuming that the water-cut, gas fraction etc. of the well remain constant.

58. If set > 0, the API tracking option is modified in gas condensate runs to conserve the surface density tracer in the oil phase only. By default the surface density tracer is transported in the oil component present in both the oil and gas phases. This option needs to be used with care, as clearly if a cell’s oil is entirely vaporized the ‘surface density tracer’ becomes infinite, giving a large negative API value.

59. Activates an alternative solver for the Extended Network option. Do not use this switch, as the solver is at present currently for experimental purposes only.

60. If set > 0, the dual permeability solver for pre-97A versions is restored. The new dual permeability solver in the 97A release is more efficient, but the old solver is still available for back-compatibility. This option switch should not be used with the CPR linear solver.

61. Not used at present.

62. Ensures back-compatibility with previous versions of ECLIPSE post-processors. Additional information on the ECLIPSE output files may not be treated correctly by pre-98A versions of GRAF and RTView, in which case setting this switch to negative produces output which can be loaded into these old versions.

63. Controls the calculation of the hydrostatic head between each connection of a multisegment well and its corresponding grid block. This depends on the difference in depth between the wellbore at the connection and the center of the grid block. The hydrostatic head is calculated using an average density of the fluid mixture in the grid block, weighted according to relative permeability so that only mobile phases contribute. The head is calculated at the start of each timestep and remains constant over the timestep.

If set > 0, the head is also averaged with its value of the previous timestep, to damp out any oscillations that may be caused by the use of explicit quantities. The averaging depends on the value n of this switch:

\[ H \text{ (used)} = \frac{(H \text{ (current)} + nH \text{ (previous)})}{(n + 1)} \]

64. Activates options for tracer modeling. Do not use this switch as it is currently intended for development purposes only.

65. If set >0, in runs using flux boundary conditions certain error conditions will be reduced to warnings to allow the run to proceed.

If set >0, in a full field DUMPFUX run, any error messages about wells connecting to multiple flux regions or local grids being situated adjacent to flux boundaries are replaced by warnings. These remain strict errors in the reduced flux USEFLUX run when wells connect both inside and outside the USEFLUX region or if any local grids are adjacent to the USEFLUX region. This switch is useful in cases with many flux regions where it is known a priori which flux region numbers constitute the USEFLUX region (using FLUXREG in the reduced run) and that there is no problem with the placing of local grids or wells in the reduced run.

If set >1, in a reduced flux USEFLUX run, any error conditions relating to local grids on the boundary of the reduced flux region are suppressed. Use this option with extreme care as the solutions are probably meaningless since the flux flows are ignored in such cases. This is only added for back-compatibility with versions prior to 98A where the boundary of the reduced region only connected to
the rest of the grid using non-neighbor connections and the error check was missing. Please do **not** use this.

66. If set > 0, it restores pre-98A functionality to Item 1 of record 2 in the **PRORDER** keyword and to the **QDRILL** option in the **WDRILTIM** keyword.

67. With effect from the 2014.1 release, this item is no longer available.

68. If set > 0, it restores the pre-98A treatment of well connection terms in the preconditioner used to solve the linear equations at each Newton iteration of the timestep. The **OPTIONS** keyword should be entered at the top of the dataset. This has no effect when using the Parallel or multisegment well options.

69. Not used at present.

70. If set > 0, it restores pre-99A definition of the **SIGMAGD** keyword. Pre-99A, the gravity drainage transmissibility was calculated using the X-permeability in the matrix, whereas post-99A the Z-permeability is used. The same effect can be achieved by setting the Z-permeabilities equal to the X-permeabilities in the matrix cells, as these data are not used elsewhere.

71. If set > 0, it restores pre-99A initialization of wells. In pre-99A versions of the code, every time certain well control keywords were called (**WCONPROD** for example), the well flows were re-initialized to zero. Now, unless the well has changed type, the well flows are not re-initialized. This has an effect on the guide rate calculation which can lead to small differences of behavior under group control; immediately after one of these keywords the well guide rate phase ratios can now be calculated from the latest flow rate ratios instead of resorting to the potential ratios.

72. Not used at present.

73. Not used at present.

Use item 4 of keyword **WVFPEXP** to prevent wells switching to THP control while they are constrained to operate on the unstable side of their VFP curve.

74. When modeling oil-wet systems with the end point scaling option activated, it is advantageous to be able to scale the water-oil capillary pressure at the maximum water saturation (using keyword **PCW**), rather than at the entry (connate) water saturation (where the water saturation equals **SWLPC**). This switch caters for this situation. Of rather less interest, but for completeness, it is possible to scale the gas-oil capillary pressure (keyword **PCG**) at connate rather than maximum gas saturation using this option switch.

If this option is non-zero and divisible by 2 then the water-oil capillary pressure is scaled at the maximum water saturation; and if it is non-zero and divisible by 3 then the gas-oil capillary pressure is scaled at the connate gas saturation.

If set to 1, then any water capillary pressure curves for which \( P_{cow} \) is entirely positive are chosen as water-wet and scaled at the entry saturation **SWLPC** while any for which \( P_{cow} \) is entirely negative are scaled at the maximum saturation **SWU**. For mixed wettability curves, it would be a bit of an illusion to arbitrarily choose which end point to scale at, so setting this option switch to 1 does nothing for such curves.

When the **SWATINIT** keyword is used to match an initial water distribution then the scaling on \( P_{cow} \) is more sensible if this option is set to 1 when the rocks are either water wet or oil wet (but no mixed wettability rocks). When the **SWATINIT** keyword is used to match an initial water distribution for mixed wettability rocks and positive capillary pressure dominated then the scaling on \( P_{cow} \) is more sensible if this option is set to 5 (minimum capillary pressure is fixed). When the **SWATINIT** keyword is used to match an initial water distribution for mixed wettability rocks and negative capillary
pressure dominated then the scaling on $P_{cow}$ is more sensible if this option is set to 7 (maximum capillary pressure is fixed). Note that setting this option to either 5 or 7 does not scale the capillary pressure above GOC, therefore, SWATINIT cannot be honored above GOC.

75. This switch is reserved for developer use - do not use.

76. If set to be non-zero, it turns off the monotonicity checks for the saturated concentration of surfactant adsorbed by the rock formation entered in the SURFADS keyword.

77. Restores back compatibility with the pre-2000A treatment of drift flux in multisegment wells.
   
   If set > 0 the pre-2000A oil-water slip model is used instead of the formula proposed by Hasan and Kabir.

   If set > 1 then in addition, in a segment inclined at an angle $\theta$ from the vertical, the drift velocity is the value calculated for vertical flow multiplied by $\cos^2(\theta)$, instead of being multiplied by $(\cos^2(\theta))(1 + \sin^2(\theta))^2$.

78. Restores the use of scratch files for pre-processing geometry data for non-neighbor connections. This may be required on systems with insufficient virtual memory to process the NNCs in memory.

79. If set > 0, the pre-2000A capillary pressure hysteresis behavior is restored; cells with an initial saturation below the wetting phase critical value uses the bounding imbibition curve. The present default behavior is to follow a scanning curve.

   If set = -1, the pre-2001A STONE1 three phase relative permeability behavior is restored in end-point scaling cases.

   **Note:** This is only applicable when switched on in the RUNSPEC section.

80. Superseded by the HYMOBGDR keyword.

81. Restores the pre-2000A ‘capping’ of the polymer concentration. If set > 0, the concentration is capped at a value of \( (\text{OPTIONS}(81) + 1) \) times the maximum injected concentration. A value of 1 restores the pre-2000A behavior of capping at twice the maximum injected polymer concentration.

82. Removes restriction on updating grid block sizes in a local grid refinement. This is only possible for use with block centered geometry.

83. Calculated values of saturations can be slightly less than 0, or slightly greater than 1 - the discrepancy depending on the accuracy of the simulation run. Although this can be useful for gauging the accuracy of the simulation it can be awkward for reporting purposes. If this flag is set > 0, values in the output files are rounded up to 0 or rounded down to 1.

84. When using the TRACTVD option the code takes mini timesteps to solve the tracer equations. ECLIPSE should calculate the correct number to take itself. If, for some reason, it is suspected that an instability in the calculation has arisen because ECLIPSE has chosen timesteps too large, the number of mini timesteps can be increased by a factor of \( \text{OPTIONS}(84)+1 \).

   If negative then the number of mini timesteps will be limited to \(-\text{OPTIONS}(84)\). This should only be used with great care!

85. Restores the pre-2005A calculation of connection voidage rates at the reference pressure for the well (see item 13 of WELSPECS). The default behavior in 2005A is that a connection’s voidage rate is calculated at the reference pressure for the fluid-in-place region of the connected grid block.

86. Write internal to natural index array to restart file.
87. Restores pre-2005A behavior of the multisegment well model with tracers.

88. The default behavior is for the emulsion viscosity model to be activated in keyword **WSEGSICD** and deactivated in keyword **WSEGAICD**. If this switch is set equal to 1 the emulsion viscosity model is deactivated in keyword **WSEGSICD**. If set equal to 2 the emulsion viscosity model is activated in keyword **WSEGAICD**. If set equal to 3 the emulsion viscosity model is both deactivated in keyword **WSEGSICD** and activated in keyword **WSEGAICD**.

89. Relax the monotonic restriction of the **ROCKTAB** keyword, allowing non-monotonic pore volume multipliers for Coal Bed Methane runs.

90. If a multiple of 2 then the closure of history matched wells with zero production rates and banned crossflow will not automatically occur.

   If a multiple of 3 then the closure of injection wells with zero injection rates and banned crossflow will not automatically occur.

   If a multiple of 5 then the derivative limiting in the solvent option (introduced in 2005A) will be turned off.

91. Restores pre-2006.1 behavior in the accumulation of the solution and residual in the linear solver. The default behavior in 2006.1 uses an improved numerical formulation (which is more stable for cells with small pore volume).

92. If set greater than zero, the default behavior of the VFP table specification in the **WCONINJE** keyword is changed so that a zero or default value indicates that the previously specified VFP table for this well should continue to be used. (The standard default here is that if a zero or default value is specified, then no THP calculations are performed.)

93. Restores pre-2006.1 well name and well list template matching behavior.

94. If set to a non-zero value, this option restores the pre-2006.1 RS extrapolation, that is a straight linear extrapolation.

95. If set greater than 0 modifies the parallel linear solver to remove connections generated to cells which are in fact completely isolated from the rest of the reservoir.

96. Using ECLIPSE releases from 2008.2 until 2013.1, the derivative of the pore volume with respect to pressure in the Palmer-Mansoori rock model (see the **ROCKPAMA** keyword) can be modified for stability. This can help if you encounter convergence problems running the Palmer-Mansoori model.

   • In 2009.1, the default behavior is that the volumetric strain effect is ignored only if it dominates the pressure effect. If a value of 1 is used, then the pre-2009.1 default behavior is restored. If a value of 2 is used, then the volumetric strain effect on this derivative is always ignored.

   • In 2008.2, the default behavior is the same as pre-2008.2. If a value of 1 is used, then the volumetric strain effect on this derivative is always ignored. If a value of 2 is used, then the volumetric strain effect is ignored only if it dominates the pressure effect.

   • From 2013.1 this switch has no effect unless **OPTIONS** item 200 is also set.

97. If set to a non-zero value, then this option restores the pre-2007.1 behavior of some sensitive constants within the program. At 2007.1, the precision of these program constants was increased to reduce cross-platform (and cross-version) differences in behavior.

98. For developer use

99. Generates warning messages from the **AQUANCON** keyword when cells cannot be connected to the aquifer because there is an active cell on the aquifer side of the current cell.
100. If set to a value greater than 0, this value is used to specify the minimum percentage separation between two consecutive Rs values for the interpolation of the PVTO tables when API Tracking is used. An improved interpolation which avoids crossing lines in the interpolated tables is also selected. The fractional separation is specified using the formula \( \text{RsSep} = 0.01 \times \text{OPTIONS (100)} \). The minimum separation criterion is then calculated as \( ((\text{Rs2} - \text{Rs1})/\text{Rs1}) \geq \text{RsSep} \), where Rs1 and Rs2 are the two Rs nodes being compared. Note that this separation should not be too large. The minimum separation is constrained to be no smaller than 0.01. If this item is defaulted or set to a negative or zero value, the minimum separation criterion between successive Rs nodes defaults to \( 10^{-7} \).

101. If > 0, this restores pre-2007.1 (and also some pre-2009.1) behavior related to well potential calculations.
   - If a multiple of 2, this reverses a 2007.1 fix in the allocation of work-space for well potential calculations. This fix only affects models which use an old-style RUNSPEC section, and it may result in a change in the reported well quantities in such models. The new behavior can also be produced with pre-2007.1 code by converting the model to use a new-style RUNSPEC section (for example, using ECLIPSE Office).
   - If a multiple of 3, this restores the pre-2007.1 logic for only calculating potentials on wells for which they are required. This is equivalent to setting the first item of the WPOTCALC keyword to ‘NO’.
   - If a multiple of 5, this reverses a 2007.1 fix in the preservation of the actual well control mode when well potentials are calculated for the prioritization option (see the PRIORITY keyword) or for prioritized drilling queues (see the WDRILPRI keyword).
   - If a multiple of 7, this reverses a 2009.1 fix in the calculation of generalized pseudo-pressure blocking factor during well potential calculations.

102. If set to a non-zero value, the pre-2007.1 implementation of the BRINE model is restored. This affects the PVT calculations in the reservoir and wells.

103. This controls aspects of the GCONSALD keyword.
   - If set to a value greater than zero value, this restores pre-2007.1 behavior.
     - In cases where a net sales, import and consumption target of zero has been specified, this sets the MAXR procedure re-injection fraction limit to a value of 1, instead of the new default value of 100.
     - It also restores the handling of the lower limit in which a non-positive lower limit was taken to mean that no lower limit was to be applied.
   - If set to a value less than zero, this sets the MAXR procedure re-injection fraction limit to \( -\text{OPTIONS (103)} \) in cases where a net sales, import and consumption target of zero has been specified.

104. If set to a non-zero value, the pre-2007.1 implementations of the scale and gas calorific value models are restored. Prior to 2007.1 the tracer concentrations used in the scale and gas calorific value calculations were lagged by two timesteps. From 2007.1 the lag has been reduced to a single timestep.

105. If set to a non-zero value, the pre-2007.1 use of group efficiency factors (specified with the GEFAC keyword) in reservoir coupling runs is restored. At 2007.1, various fixes were made to this part of the code.

106. If set to a non-zero value, the pre-2007.1 gravity drainage behavior is restored. At 2007.1, a bug was fixed which would may a change in results for models that contain the GRAVDR keyword.
107. If set to a non-zero value, the pre-2007.1 behavior when the INTPC option for integrated capillary pressure is used with the alternative gravity drainage model specified by GRAVDRM is restored. Although INTPC was previously allowed with GRAVDRM, the pre-2007.1 behavior did not lead to the correct final recovery in this case, whereas the 2007.1 behavior should do so.

108. If set to a non-zero value, the pre-2008.1 behavior for scale deposition is restored. Scale deposition will only be applied to producing completions (which could belong to producers or crossflowing injectors).

109. If set to a non-zero value, the pre-2007.1 behavior of the RPTONLY and RPTONLYO keywords in the SUMMARY section of restarted runs is restored. At 2007.1 this behavior was changed so that these keywords were ignored if they were present in the SUMMARY section in restarted runs, and they had to be re-specified at or after the restart time in the SCHEDULE section.

110. If set to a non-zero value, the pre-2007.3 behavior of the AQUCHWAT keyword is restored. The density calculation for the water in the aquifer, which is then used in the aquifer inflow equation, has been altered to incorporate the aquifer reference pressure.

111. If set to a non-zero value the pre-2008.1 behavior of the WTEST keyword is restored. This causes the WTEST keyword to reset for wells closed as part of a group of producers closed by a single event.

112. If set to 1 the pre-2008.1 behavior of the RADFIN4 keyword is restored. The 2008.1 behavior should lead to the porosities always being assigned correctly to the LGR cells, whereas the pre-2008.1 behavior can be incorrect for the case where THETA=4.

113. If set to 1 the pre-2008.1 behavior of API tracking in local grid refinements is restored. The 2008.1 behavior correctly calculates oil density in global grid cells using an oil-in-place weighting of local grid cell values.

114. If set to a non-zero value, an improved water and solvent saturation bound-checking is performed.

115. If set to a non-zero value the pre-2008.1 behavior of analytic aquifer inflow equations is used. The 2008.1 behavior correctly calculates the flow from the reservoir to the aquifer in the case of connate water/gas. This change affects all analytical aquifers.

116. This option affects the POLYMER or FOAM models with the shear thinning option for releases between 2008.2 and pre-2011.1. If set to non-zero value in these releases, it reverts to pre-2008.1 behavior, which is now the default behavior (upstreamed polymer/foam concentration in the injecting well).

117. Affects dual porosity and dual permeability values using the MULTNUM, FLUXNUM, PINCHNUM or OPERNUM keywords. If this option is 1, then MULTNUM regions must be specified separately for matrix and fracture. If this option is 2 or greater, then regions specified by all of these keywords must be specified separately for matrix and fracture.

118. If set to a non-zero value the pre-2008.2 behavior for the treatment of the DRSDT or DRVDT keywords in the presence of vaporized oil and dissolved gas respectively is restored; that is, the limits specified by these keywords are ignored in the presence of vaporized oil and dissolved gas.

119. If set to a non-zero value the pre-2009.1 capillary pressure calculations for the surfactant model are restored. See "Surfactant model" in the ECLIPSE Technical Description.

120. If set to 1 the pre-2008.2 convergence criterion for the standard solver used for tracer tracking is restored.

121. If set to a non-zero value the pre-2008.2 behavior for the calculation of relative permeabilities using Stone’s Second Method (STONE2) with end-point scaling is restored.

122. If set to 1, the pre-2008.2 polymer dead pore volume model is restored.
123. This option affects the behavior of Carter-Tracy aquifer influence function tables. In 2009.1, the interpolation of influence function tables has been improved. The handling of extrapolated data was modified in 2009.2. In 2010.1, the default Carter-Tracy influence table has been extended to cover larger dimensionless time values. If set to 1, the pre-2009.1 behavior is restored. If set to 2, the 2009.1 behavior is restored. If set to 3, the 2009.2 behavior is restored.

124. If set to a non-zero value the pre-2009.1 calculation of the summary quantities FTPC, GTPC, WTPC, CTPC, FTIC, GTIC, W Tic and C Tic for concentrations of partitioned tracers will be restored.

125. If set to a positive value $N$, this changes the tolerance used when checking whether the current simulation time is a report time to a new tolerance of $10^{-N}$. The default tolerance is $10^{-3}$ which is equivalent to setting this 3.

126. If set to 1, all active tracers will be solved in each Newton iteration instead of at the end of a timestep. This option can be used to reduce time lag in tracer dependent properties. If set to 2, all active tracers will be solved at the end of a timestep. The default behavior is to solve all tracers at the end of a timestep, unless ECLMC is specified in the RUNSPEC section, in which case the default behavior is to solve Brine, Surfactant and Alkaline tracers in each Newton iteration, but all other tracers at the end of a timestep.

127. If set to a positive value, the pre-2009.1 behavior for tracer adsorption, using a non-cascade tracer solver, is restored. Prior to 2009.1 the adsorbed tracer amount was not properly reset following a convergence failure.

128. If set to a non-zero value, the pre-2009.1 behavior for the calculation of residual oil saturation for the Stone I relative permeability model with end-point scaling is restored. Prior 2009.1, in some circumstances, the critical saturations used in the calculation of the residual oil were not subject to end-point scaling.

129. For developer use.

130. If set to a non-zero value, the pre-2009.1 behavior for the specific heat unit conversion used in the temperature option is restored.

131. Except for the MISCNUM keyword, the documentation states that all REGIONS section keywords should be assigned a minimum value of 1. However, it is possible that pre-processor or manual edits to a keyword definition, can set a value of 0, even for inactive cells. To be compatible with the treatment of REGIONS section keywords using EQUALS / ADD / MULTIPLY keywords, if REGIONS data is specified using a REGIONS keyword directly, then error messages can be output for both active and inactive cells. To do this, set this switch to a non-zero value.

132. If set to a non-zero value the pre-2009.1 behavior for WLIFT is obtained. Prior to 2009.1 a well would only be revived once when it died on THP control by increasing its Artificial Lift Quantity even if the maximum ALQ value had not been reached.

133. If set to a non-zero value the guide rate calculation method of ECLIPSE 300 is used. Well and group guide rates are allowed to change for the first NUPCOL Newton iterations instead of remaining fixed throughout the timestep.

134. If set to 1, a wider range of global cells is used as reference when calculating the geometry of LGR cells. This can provide better output to the EGRID file for visualization purposes.

135. If set to a positive value, the pre-2009.1 interpolation for the SOF32D saturation functions family is restored.

136. If set to a non-zero value, the pre-2009.1 behavior for the decay of an adsorped tracer using a non-cascade tracer solver is restored.
137. If set to a non-zero value, the pre-2009.1 behavior for inter-block velocity calculations is restored.

138. If set to a non-zero value, the pre-2009.1 behavior for the modification of relative permeabilities using DPKRMODE in dual porosity models is restored.

139. If set to a multiple of 2, the switch activates an alternative method for calculating the mixture density in stagnant connections at the bottom of the well. Instead of using average well phase fractions to estimate the density, it is set equal to the density at the lowest open connection. If set to a multiple of 3, the switch forces crossflowing connections to use the segmented density calculation method instead of the average calculation. This option is only applicable to non-friction wells using the SEG density calculation method (see item 12 of WELSPECS). This option can be useful in wells with large density variations between the uppermost and lowermost connections where the AVG density calculation method can be inappropriate. However, it should only be used during problem periods and then be switched off otherwise as it can cause convergence problems.

140. If set to a non-zero value, the pre-2009.2 behavior for the interpolation of PVTO tables for API tracking is restored. For the 2009.2 version onwards the interpolation procedure has been modified to avoid duplicate Rs nodes which could occur for some combinations of table data and minimum Rs node spacing.

141. If set to 1, the pre-2009.2 behavior for the modification of non-neighbor connections in cells which have been coarsened is restored. For the 2009.2 version onwards, the handling of NNCs across x- and y-faults is much improved, and the transmissibility of all NNCs within coarsenings may be slightly altered.

If set to 2, the pre-2011.1 behavior for the transmissibility calculations for coarse cells in which some fine cells are active and others inactive is restored. For the 2011.1 version onwards, a single inactive fine cell in a coarse cell will not zero the transmissibility for the entire coarse cell.

This option should only be used to confirm that a particular change in behavior is due to the improved modeling in 2009.2 and later.

142. Reserved for future use.

143. If set to a non-zero value, this restores the pre-2010.1 behavior for the reporting of partitioned tracer flowrates in river reaches.

144. If set to a non-zero value, the pre-2009.2 behavior for calculating flow rate limits for the WELDRAW keyword is used.

145. As explained in item 12 of WELSPECS, the hydrostatic heads calculated using segmented densities in the wellbore are damped to reduce the likelihood of stability problems. This is done by combining values from the previous timestep with those of the current timestep in a weighted average. An error has been identified in the implementation of this damping which meant that more damping was being applied than intended. This error has been fixed in the 2010.1 version of the simulator but has not been made the default behavior as it may result in changes in simulation results. The correct damping can be applied by setting this item to a non-zero value.

146. If set to a non-zero value, fluid in place reports and pressure averages include contributions from excavated blocks.

147. If set to a non-zero value, the pre-2010.1 behavior is restored for relative permeability interpolation based on capillary number in the Surfactant Model. For the 2010.1 version onwards the interpolation between immiscible and miscible relative permeability curves is only performed for blocks with a non-zero surfactant concentration. The immiscible curves are used for blocks with a zero surfactant concentration.
148. If set to a non-zero value, the pre-2010.1 behavior of river infiltration equations in the unsaturated regime is restored.

149. If set to a non-zero value, the pre-2010.1 behavior is restored for solving tracers in combination with multisegment wells (MSWs). For the 2010.1 version onwards, an error has been fixed in the coupled iterative solution of the tracers which previously led to wrong tracer rates and concentrations in the MSW segments. The change affects single phase, two phase and partitioned tracers.

150. If set to a non-zero value, the pre-2010.1 behavior is restored for GECON item 8. For the 2010.1 version onwards, the simulation will not be stopped if a group includes a follow-on well due to be opened in the next timestep.

151. If set to a non-zero value, the pre-2010.1 behavior is restored for calculating pinchout transmissibility values when item 4 of the PINCH keyword is set to ALL. This had an effect only in the 2010.1 and 2010.2 releases. From 2011.1 onwards the pre-2010.1 behavior has been restored unconditionally.

152. If set to a non-zero value, the pre-2010.1 behavior for the RPTRST argument RFIP is restored. The pre-2010.1 behavior calculated reservoir fluid-in-place to be surface fluid-in-place multiplied by formation volume factor. From 2010.1 this is calculated as pore volume multiplied by saturation.

153. Reserved.

154. If set to 1, the pre-2011.1 behavior of the tracer-tracking option (TRACERS) is activated. This affects how the injected concentration for crossflowing wells is calculated.

155. This item relates to protection against bad cross-processor NNC set-up

   • If set to 1, a BUG message protecting against bad set-ups of cross-processor NNCs is downgraded to a WARNING message. This may be used to allow some cases to run which would otherwise terminate with a BUG message. However, this is not recommended as the BUG message does indicate a serious problem.

   • If set to 2, all the effects of setting to 1 are introduced. Furthermore, the pre-2010.2 behavior regarding NNCs spanning multiple processors in parallel is restored. This behavior is known to be problematic in certain cases.

156. If set to a non-zero value, the pre-2010.2 method for end-point scaling is used with the minimum oil saturation tables (SOMGAS or SOMWAT).

157. If set to 1, the pre-2011.1 handling of SUMMARY keywords AAQR, AAQT, ALQR and ALQT is restored. Before 2011.1, the list associated with these keywords could contain an element referring to a gas analytic aquifer. If the list was empty, all analytic aquifers including gas one were listed. From the 2011.1 version on, using these keywords with a gas analytic aquifer is not allowed. Additionally, the ALL summary keyword implies the keywords AAQRG and AAQTG.

158. If set to 1, the phase saturations and oil-gas ratios are limited to explicitly bounded values. It is possible for a non-linear iteration to produce saturation values that are outside the physical range between 0.0 and 1.0. Usually the amount by which they are outside this range is small, but to prevent any problems with physical correlations, setting this option to 1 forces all saturations (for water, gas, oil and solvent) to be zero if less than zero, and to be one if greater than one, at the end of each non-linear iteration. Similarly, oil-gas ratios will be forced to zero if they are negative.

159. Reserved.

160. This item relates to changes to looped MSWs and ICDs. Pre-2011.1 behavior may be obtained by setting this item to 42 (that is the first, second and fourth bullet points below). The item has the following behavior:
• If set to a multiple of 2, the pre-2011.1 looped MSW behavior is restored. This principally alters the MSW solution predictor.

• If set to a multiple of 3, the pre-2011.1 behavior of item 6 of WSEGSOLV is restored. This affects the range of linearization for ICDs, AICDs, labyrinth devices and valves in looped MSWs.

• If set to a multiple of 5, an alternative MSW linearization is applied for looped wells for high segment flow rates.

• If set to a multiple of 7, the pre-2011.1 ICD solver implementation is restored.

161. If set to 1, the pre-2011.1 miscible viscosity and density calculations are restored.

If set to 2, the 2011.1 miscible viscosity and density calculations are restored.

The default miscible viscosity and density calculations are similar to the original pre-2011.1 calculations.

162. If set to 1, the pre-2011.1 behavior is restored for the reporting of the scaled table end-points in the debug file when EPSDBGCS or EPSDEBUG is specified. For the 2011.1 version the scaled values are reported when no scaling is applied.

163. If set to 1, duplicated records in the FAULTS keyword are ignored. If a fault is specified twice, any multipliers associated with it are applied twice.

164. If set to 1, the pre-2011.1 behavior is restored when reading the SURFVISC keyword. From 2011.1 onwards the water viscosity in SURFVISC at zero surfactant concentration and reference pressure must be equal to the value entered in the PVTW keyword.

165. If set to 1, the pre-2011.1 behavior for initial tracer concentrations in cells containing no host phase is restored. This switch also activates equivalent behavior for the initial concentrations of surfactant, foam and polymer.

166. If set to a 1, the pre-2011.1 behavior is restored for the modeling of the two-phase relative permeability of oil to gas when using the EHYSTR or EHYSTRR keywords with item 2 set to 4 (Killough hysteresis applied to both wetting and non-wetting phases). In some cases it has been found that the two-phase relative permeability of oil to gas can exceed the maximum value specified in the drainage and imbibition oil saturation function tables, or the maximum value specified via the end-point scaling keywords KRO and IKRO (and their directional and irreversible variants) if ENDSSCALE has been specified. Depending upon the choice of three-phase relative permeability model, this can result in the three-phase oil relative permeability also exceeding the maximum value from the tables or end-points. This has been addressed for the 2011.1 version by constraining the maximum value of the two-phase relative permeability of oil to gas to the maximum of the values specified in the drainage or imbibition tables, or end-points. The two-phase oil relative permeability of oil to gas is combined with the two-phase oil relative permeability of oil to water to form the three-phase oil relative permeability. The constraint described here, therefore, will apply to all three-phase oil relative permeability models available in ECLIPSE 100.

167. Reserved.

168. If set to 1, the pre-2011.1 behavior of WDFACCOR is recovered. This changes D-factor calculations for flow-dependent skin factor if fewer than 3 phases are present in reservoir.

169. If set to 1, the pre-2011.1 behavior is restored for the treatment of the NOHYST keyword. Prior to 2011.1, hysteresis was not deselected by this keyword if specified after the SATOPTS keyword or if WAG hysteresis was specified via the WAGHYSTR keyword.
170. If set to 1, this deactivates the processing of the ACTNUM data specified in an EGRID file identified by the GDFILE keyword. Note that in previous versions, the default was not to process ACTNUM; from 2011.1, processing ACTNUM is the default.

171. If set to a negative value, the threshold tolerance for the adsorption calculation when using the cascade tracer solver is changed. The threshold tolerance affects the adsorption by skipping the calculation in cells where the amount of tracer and the rock mass is less than the threshold value. Item 171 can be used to control the threshold value. If set to a negative value, the threshold value is taken as \(10^N\) where \(N\) is the value specified in item 171. The default value is \(10^{-12}\). The pre-2011.1 value can be restored by setting \(N\) to -6. This option affects active tracers, such as surfactant, alkaline and foam, as well as environmental tracers solved by the cascade method.

172. If set to 1, the pre-2011.1 behavior of GSSCPTST is recovered. This affects simulations with local grid refinements.

173. If set to 1, the pressure is chopped if it falls below 1 atmosphere.

174. If set to 1, the pre-2011.1 behavior is restored for the treatment of the SWATINIT keyword when read via the IMPORT keyword. Prior to 2011.1, this method of reading the SWATINIT keyword resulted in some of the saturation table data consistency checks being omitted.

175. Reserved.

176. If set to 1 this reverts to pre-2012.1 behavior for polymer and brine in standard wells.

177. This option allows the use of the gradient option in parallel runs. This combination of features is unsupported and so should be used with caution as it may lead to unexpected results or errors.

178. If set to 1, the pre-2011.2 behavior for the processing of JFUNCR with hysteresis is restored. Prior to 2011.2 the per region processing of J-functions was only applied to the imbibition saturation tables for hysteretic models.

179. If set to 1, the pre-2011.2 behavior for calculating radial velocities is restored.

180. If set to 1, this restores the pre-2012.1 behavior for table look up for the coal bed methane option, using SATNUM and PVNUM instead of COALNUM. This affect the keywords LANGMUIR, LANGSOLV, COALADS, COALPP and DIFFCOAL.

181. Not used.

182. This switch reverts part or all of the looped multisegment well (MSW) solver to the pre-2012.1 behavior.
   - If positive and divisible by 2, the looped MSW solver will not resolve unconverged linear solutions using a direct solver. This may speed up the simulation, but may lead to failed well solutions.
   - If positive and divisible by 3, the predictor will not set every segment to a non-zero flow. This can lead to a singular well Jacobian and therefore problems with the well solution.
   - If positive and divisible by 5, the pre-2012.1 behavior of the tracer solver with looped multisegment wells is recovered. This behavior change can only be made in the RUNSPEC section.

183. When set to 1, this switch reverts the network balancing option to pre-2012.1 behavior. An additional check was added to the network balancing option to ensure that all well THP targets reflected their parent group’s node pressure. This check improves the accuracy of network balancing in certain circumstances such as when the reservoir coupling option is in use.
184. This switch reverts the behavior of hysteresis to pre-2012.1 behavior. It may affect two separate behaviors:

- When set to a positive value divisible by 2, this switch reverts the enumeration of reservoirs with hysteresis to pre-2012.1 behavior. This alters the initial \( R_v \) value in the reservoir to take account of capillary pressure.

- When set to a positive value divisible by 3, this switch reverts the calculation of critical gas saturation used to determine saturation chops to the pre-2012.1 behavior. Prior to 2012.1 the saturation curves were sometimes taken from the wrong region for this calculation.

185. This switch allows restarting from versions before 2012.1 if the combination of tracers with polymer or solvent or GI-condensate has been used. In this case, older versions of the code may have corrupted the data. Use of this switch is not recommended.

186. If set to 1, this switch reverts the \texttt{WCONPROD} and \texttt{WCONINJE} keywords to their pre-2012.1 behavior with respect to initialization for multisegment wells. Prior to 2012.1, any change of control mode by \texttt{WCONPROD} or \texttt{WCONINJE} caused the first guess of the well solution to be re-initialized. From 2012.1, this is not the case unless this option is set. This affects multisegment well convergence characteristics.

187. When set to 1, this switch reverts partitioned tracers to pre-2012.1 behavior. Prior to 2012.1, injecting connection flows of tracer were incorrectly calculated during the solving of the tracer equations.

188. Reserved.

189. When set to 1, this switch reverts the behavior of GSG non-neighbor connection and transmissibility allocation to pre-2012.1 behavior. This behavior in parallel can cause, in some situations, the non-neighbor connections and transmissibilities to be incorrectly communicated between processors.

190. If set to 1 (in the \texttt{SOLUTION} or earlier sections), the pre-2013.1 method of checking input enumerated saturation data is selected. For the 2013.1 version, the checking of input enumerated saturation data has been extended to check the data values against their expected end-points for end-point data derived both from saturation tables and end-point scaling arrays and tables.

191. This item has been deprecated. To control the number of messages output for dead block connections use item 13 of the \texttt{MESSAGES} keyword or \texttt{OPTIONS} item 220.

192. This switch controls the number of warnings output regarding linear convergence problems. It can restore pre-2013.1 levels or suppress message output further.

- If set to 1, this switch restores the pre-2013.1 verbosity of messages regarding linear equation solver convergence problems

- If set to -1, it reduces messages regarding linear equation solver convergence problems further.

193. If set to 1, the pre-2013.1 behavior of approaching report steps will be active when using the \texttt{ZIPPY2} keyword.

194. If set to 1, the pre-2012.2 behavior for assigning saturation end-points from end-point scaling arrays for two-phase models will be restored.

195. If set to 1, the pre-2012.2 behavior for a zero determinant detected in the linear solver is restored (stopping with an error). The new behavior is to chop the timestep and try to recover.

196. This option should not be needed in 2013.1 and later releases.

- If set to 1, the pre-2012.1 treatment of network choke branches will be used.
197. If set to 1, a modified WAG hysteresis gas relative permeability model will be used in situations where the transformed gas saturation at the start of the secondary or subsequent drainage cycle falls below the gas saturation at the start of the previous drainage cycle. This modification prevents excessive falls in gas relative permeability which can occur in such circumstances when the primary drainage curve is used to construct the secondary drainage cycle. More details are provided under "Hysteresis in WAG floods" in the ECLIPSE Technical Description.

198. If a well is set not to allow crossflow (WELSPECS item 10) then, if there are convergence problems, this option will close any open connections that are trying to back flow and retry the well Newton iteration. This option should be set to the number of re-tries the simulator can perform before moving onto the next reservoir Newton iteration. The option is not enabled for multisegment wells.

199. If set to 1, the pre-2013.1 treatment of directional rock compression transmissibility multipliers will be used. This uses the x-direction multiplier for all NNCs.

200. If set to 1, the pre-2013.1 calculations for ROCKPAMA will be used. The derivative changes as specified in OPTIONS item 96 are ignored by default from 2013.1.

201. If set to 1, the additional WAG hysteresis regions specified via the WH2NUM and WH3NUM keywords and the associated additional processing (which applies whether these have been specified or defaulted) will be ignored. This should be specified in the RUNSPEC section. This option is only relevant when the WAGHYSTR keyword has been specified.

202. If set to 1, the pre-2013.1 version of the debug output for the scaled saturation tables specified via either the EPSDEBUG and EPSDBGS keywords will be selected.

203. If set to 1, this enables LGRs in USEFLUX runs where one or more parent cells are inactive because they lie outside the USEFLUX regions. Note that, by default, such LGRs are disabled.

204. If set to 1, the pre-2013.1 behavior is restored for the model initialization process for the case where item 2 of the EHYSTR keyword is set to -1.

205. If set to 1, connate gas will be included in the two-point scaling of the oil relative permeability to gas and water; that is, the maximum attainable oil saturation will be assumed to be constrained both by connate water and gas. By default, the connate gas is omitted from these calculations.

206. If set to 1, the pre-2013.1 behavior is restored for well connection ordering. This affects some cases with COMPORD and WRFT or WRFTPLT.

207. If set to 1, the pre-2013.1 behavior is restored for tracer calculations in cases with PARALLEL, VAPOIL, DISGAS and non-neighbor connections. Note that we advise against the use of this option as it recovers incorrect behavior in some cases.

208. If set to 1, the pre-2013.1 behavior is restored for error-checking of the second argument in keywords PLYVISCS, PLYVISCT and PLYVSCST. In particular, a lower bound of unity is now applied by default; activating this switch will allow any positive value.

209. If set to 1, the pre-2013.2 behavior is restored. The new, default behavior is to use the maximum adsorbed polymer concentration itself in a cell as the reference point for allowing (increasing adsorbate) or disallowing (decreasing adsorbate) changes in the adsorbed concentration. The previous behavior used the maximum solution polymer concentration as a proxy for this and did not take into account salinity changes.

210. If set to 1, the pre-2014.1 behavior is restored for the calculation of relative permeabilities when using the SURFACT or LOWSALT models with ENDScale.

211. If set to 1, the pre-2014.1 behavior is restored for the calculation of the capillary number when using the SURFACT and DUALPORO models.

Keywords

1545
212. If set to 1, connate gas will be included in the end-point scaling calculation of the saturation at which \texttt{KRWR} occurs in the end-point scaled saturation space.

By default connate gas is omitted from these calculations.

213. If set to 1, the pre-2014.1 behavior is restored for the calculation of water relative permeability used as a component in the calculation of the three-phase oil relative permeability for Stone's Second Model \texttt{STONE2}.

For the 2014.1 version, corrections have been made to the calculation of water relative permeability when subject to hysteresis and end-point scaling.

214. If set to 1, the pre-2014.1 behavior is restored for the calculation of the component cell heights for the \texttt{GRAVDR} model when used with either the \texttt{SURFACT} or \texttt{LOWSALT} models.

215. If set to 1, unset values for enumerated initial pressures in coarsened grids will be replaced with the value from an adjacent cell.

If set to 2, unset values for all enumerated initial conditions in coarsened grids will be replaced with the value from an adjacent cell.

This can be used to avoid issues where a pre-processor is unable to assign an enumerated value to an inactive cell which is used as the representative cell for its coarsened cell.

216. If set to 1, the error emitted in parallel when a well is defined using \texttt{WELSPEC} on an LGR that is switched off is downgraded to a warning.

217. If set to 1, the pre-2014.1 behavior for \texttt{VDFLOW} is restored such that the value of the inertial coefficient for non-Darcy flow is only applied to the first saturation region when more than one saturation regions have been defined. For the 2014.1 version, the inertial coefficient is applied to all saturation regions.

218. If set to 1, the pre-2014.1 behavior is restored for the non-Darcy flow component specified via the \texttt{VDFLOW} or \texttt{VDFLOWR} keyword. Prior to 2014.1 this was only applied to the phase mobility calculations for the well connections if the \texttt{FHRECBL} keyword had also been specified.

For the 2014.1 version, by default, the non-Darcy flow component due to the \texttt{VDFLOW} or \texttt{VDFLOWR} keyword is not applied to the well connections in any circumstance. If the \texttt{VDFLOW} or \texttt{VDFLOWR} flow component is required to be applied to the well connections, this may be enabled by setting this \texttt{OPTIONS} item to 2. Wells or well completions for which D-Factor calculations have been specified via the \texttt{WDFAC} keyword the \texttt{WDFACCOR} keyword or item 12 of the \texttt{COMPDAT} keyword (or item 13 of the \texttt{COMPDATL} keyword) will supersede those due to the \texttt{VDFLOW} or \texttt{VDFLOWR} keyword.

219. If set to 1, the pre-2014.1 message output format for parallel runs is used; messages from slave processors will not appear in the output files for the master process.

220. If set to 1, the message output limit from \texttt{MESSAGES} item 13 is ignored.

221. If set to 1, the pre-2014.1 behavior for determining when a non-neighbor connection should be deleted based on its transmissibility is restored. Additionally, values which are below the cutoff value for all of transmissibility, diffusivity and thermal transmissibility will be preserved provided that the NNC itself is preserved. Note that, if this option is used, the diffusivity and thermal transmissibility settings from the \texttt{MINNCT} keyword will have no effect.

222. If set to 1, the pre-2014.1 behavior for Killough hysteresis model for water wet systems will be selected. For the 2014.1 version, corrections have been made to the calculation of the wetting phase relative permeability when subject to hysteresis and end-point scaling.

223. If set to 1, more than one occurrence of the \texttt{GRIDOPTS} keyword is permitted.
224. If set to 1, the pre-2014.1 behavior for CECON is restored. UDAs used in CECON are applied to all connections in the wells specified with the particular UDA.

225. If set to 1, the pre-2014.1 versions of the LOWSALT and SURFACTW options are restored. The primary differences between the 2014.1 and pre-2014.1 versions relate to the modeling of end-point scaling when the ENDScale option has been specified. Prior to the 2014.1 version, end-point scaling was only applied to the high salinity end-points using SWL and associated keywords. For the 2014.1 version, end-point scaling has been added for:

- the low salinity (oil-wet) saturation tables by introducing a new set of end-points including LSWL
- for the high salinity (water-wet) saturation tables by introducing a new set of end-points including HWSWL
- for the low-salinity (water-wet) saturation tables by introducing a new set of end-points including LWSWL

If set to 1, the new end-point scaling arrays may be specified but they will not be used in the simulation.

226. If set to 1, the effective well bore radius, used in the calculation of the well productivity index, can be greater than the drainage radius. Otherwise, the effective radius will be taken as the minimum of the effective radius and the drainage radius when calculating a well's productivity index. For the ECLIPSE 300 equivalent please refer to item 317 of the OPTIONS3 keyword.

227. If set to 1, the preferred phase of an injection well is not enforced to match the actual injection phase. For the ECLIPSE 300 equivalent please refer to item 318 of the OPTIONS3 keyword.

228. By default, prior to polymer shear calculations, ECLIPSE 100 now includes the mobile water saturation in the conversion from flow rate to velocity and, for runs including keyword SHRATE, applies a modified conversion factor from velocity to shear rate. If this item is set to 1, this activates the pre-2014.1 conversion from flow rate to velocity in which the mobile water saturation is excluded and may be applied to models that use either PLYSHEAR or PLYSHLOG.

If set to 2, this activates the pre-2014.1 conversion from velocity to shear rate and may only be applied to models that use keyword SHRATE.

If set to 3, this activates the pre-2014.1 conversions from flow rate to velocity and from velocity to shear rate and may only be applied to models that use keyword SHRATE (see "Polymer flood model" in the ECLIPSE Technical Description for details of the forms of these conversions).

229. If set to 1, the pre-2014.1 behavior for Killough hysteresis model for oil-wet systems will be selected. For the 2014.1 version, corrections have been made in the two phase oil to water relative permeability and connate water saturation was included in the calculation of oil to gas relative permeability.

230. If set to 1, the pre-2014.1 behavior of the 'TRNC' timestepping control for POLYMER/BRINE and SOLVENT runs will be restored.

231. If set to 1, the pre-2014.1 behavior of inverse lookup with three-point end-point scaling will be selected. For the 2014.1 version, corrections have been made in cases where SCALECRS is activated.

232. If set to 1, DUMPFLUX and USEFLUX may be used with various CEOR and tracer options which are normally banned in this combination, but it is the user's responsibility to ensure that no CEOR or tracer effects occur outside the flux boundary.

233. If set to 1, the pre-2014.1 behavior for the input processing of the two-dimensional capillary pressure data specified via the PCG32D and PCW32D keywords will be restored. Prior to 2014.1, only the first table was used for all saturation regions. If more than table was specified, the second and subsequent
tables were ignored. For the 2014.1 version, successive tables are read and allocated to their associated saturation regions.

234. If set to 1, the pre-2014.1 behavior for the scaling of gas relative permeability at residual saturation of the displacing phase (KRGR) for gas-water models with end-point scaling will be restored. Prior to the 2014.1 version, the residual saturation used to determine the gas relative permeability was calculated using the critical oil in water saturation. For the 2014.1 version, this is calculated using the critical water saturation.

235. If set to 1, the printed grid property tables in the PRT file will be scaled so that the maximum value in the table is always between 1 and 10. The units are appropriately rescaled to give the correct value overall. The default value of 0 gives the pre-2014.1 scaling, which only allowed certain rescaling magnitudes.

236. If set to 1, the pre–2014.1 geometry calculations are used to calculate neighbor transmissibilities for corner point geometry. Prior to the 2014.1 version, in some extreme cases (principally with a perfectly vertical fault which has cells on both sides sloping towards the fault) a small transmissibility value can be calculated between cells which are neighbors in i,j,k space even though the cells do not physically touch. When this switch is used, these cell indices and transmissibility values are also output to the debug file.

237. If set to 1, brine injection with the WSALT keyword may be used in dual porosity models with local grid refinements. This combination is banned otherwise. It is the user's responsibility to ensure that the local grid properties do not differ from global cells.

238. If set to 1, the pre-2014.1 behavior of the default interphase threshold pressure calculation will be activated. For the 2014.1 version, the interphase threshold pressure when defaulting item 3 of the THPRES keyword, will be calculated from the initial conditions considering any phase present in the upstream cell.

**Examples**

To use items 21 and 34.

```plaintext
OPTIONS
20* 2 12* 1 /
```

or

```plaintext
OPTIONS
20* 2 /
OPTIONS
33* 1 /
```

**Keywords**

1548
Options3 Activates special program options

The OPTIONS3 keyword can be used to activate special options within particular facilities in ECLIPSE 300. These options are mainly of a temporary or experimental nature, or act to restore back-compatibility with earlier versions of the code.

The keyword should be followed by integers, each of which activates a special option within particular ECLIPSE 300 facilities. A value equal to zero switches off the special option. A value other than zero activates a special option. The data field should be terminated by a slash (/). If fewer than the maximum number of integers are read, the remainder are left unchanged. Once an option is activated, it remains activated and retains its value in all subsequent uses of OPTIONS3 keyword unless it is explicitly changed or turned off. Note that repeat counts (for example 3*0) can be used if required (there must be no spaces before or after the asterisk).

1. Set to 1 to obtain the explicit hydrostatic head values in the well model when the head calculation is based on an averaged wellbore density.
   
   Set to -1 to obtain the implicit hydrostatic head values in the well model when the head calculation is based on an averaged wellbore density.

   By default, ECLIPSE 300 uses a head calculation that is based on segmented wellbore densities (see item 12 in WELSPECS keyword). This head is always treated explicitly and so this switch will have no effect.

2. Set to 1 to obtain the FREEZEPC option. This will be overridden if the FREEZEPC, NOFREEZE or DPCDT keywords are used.

3. Set to 1 for saturation clipping

4. New Material Balance Control on Convergence
   
   In thermal runs (2006.1 onward) this option is automatically turned on. It can be turned off in thermal runs by setting a negative value.

5. Set to 1 to use a saturation pressure test to determine the phase in separator calculations.

6. With effect from the 2014.1 release, this item is no longer available.

7. Set to 1 to interpolate initial compositions from the constructed internal tables rather than directly from the user data.
   
   This is to obtain back-compatibility with the 95 versions.

8. Set to 1 to add the water-gas capillary pressure to the water phase rather than to the gas phase for the Gas-Water model.

9. Set to non-zero value to assign a zero THP for zero rate wells. Set to -1 to obtain the pre-2009.1 behavior for this option switch.

10. Controls output of VFP extrapolation messages
    
    0 Report VFP extrapolation at end of run
    
    1 Report VFP extrapolation at end of each report step
    
    2 Report VFP extrapolation at end of each report step and stop run if extrapolation has occurred.

11. If > 0 potential reversal chops are switched on.

Keywords

1549
If =2 then the potential reversal chops only apply if a significant mobility contrast exists in the two cells.

12. If > 0 the pre-98A equilibration method is used.

13. If > 0 the pre-98A average specific volume derivative is used to normalize the solution change reported in the non-linear iterations.
   If < 0 the post-98A average specific volume derivative is used to normalize the solution change reported in the non-linear iterations.
   By default, ECLIPSE 300 uses the post-98A method except when running the Thermal option, which uses the pre-98A method by default.

14. If > 0 the actual specific volume normalization is used instead of the average specific volume derivative in the non-linear iterations.

15. If > 0 the reference surface tension, if defaulted, is taken as the initial cell-by-cell values rather than the field average. This was the pre-98A behavior.

16. If > 0 use pre-98A constant slope extrapolation of K-values.

17. If = 1 switch off well BHP solution damping.

18. If = 1 disable the saturation-based Appleyard Chop for the Thermal option.
   If = -1 use pre-2004A saturation-based Appleyard Chop for the Thermal option.

19. Turn off non-linear speed-up due to Gain.
   If =1, turn off Gain when there has not been a change of state in any cell.
   If =2, turn off Gain.
   Note that this is also affected by the setting of Item 68. Both items must be set to ‘on’ for the non-linear speed-up due to Gain to be used (so, by default, Gain is generally off).

20. If > 0 use the COLSUM constraint in the Nested Factorization preconditioner.
   If < 0 use the ROWSUM constraint in the Nested Factorization preconditioner.
   (By default, COLSUM is used in fully implicit and ROWSUM in IMPES and AIM cases. Prior to 99A COLSUM was used in all cases.)

21. If > 0 attempts to stop Newton iterations oscillating between alternate states.
   If < 0 the damping of oscillations in Newton iterations is not activated.
   By default, ECLIPSE 300 does not attempt to damp any oscillations in the Newton iterations, except when running the Thermal option, which damps these oscillations by default.

22. If > 0 causes the program to use an alternative set of tables from Haywood instead of the tables from Perry and Green (Thermal option only).

23. If > 0 normalize residuals by total moles rather than hydrocarbon or water moles.

24. If < 0 read the 6 column PVCO keyword but ignore the viscosibility (the pre-2002A behavior)
   If > 0 use pre-99A version of PVCO with only 5 columns.

25. If > 0 disable the Appleyard Chop for compositional mode.

26. Activates the new Rachford-Rice solver in the flash (compositional mode).
1. Vector flash
2. Vector and scalar flash

27. If = 1 disable temperature-based Appleyard Chop for the Thermal option.
28. If > 0 use constant hydrostatic head for well calculations.
29. If > 0 use scalar flash to converge those reservoir cells which are making the transition from a single-phase state to two hydrocarbon phases. This switch restores compatibility with the pre-99A treatment.
30. Controls how advance gas is to be used during reinjection. If:
   =1 uses the mixing tank treatment where the advanced gas is mixed with the produced gas and gas from this mixing tank is reinjected. The advanced gas rate is added to the produced gas rate to form an available gas rate for reinjection subject to fuel and sales gas commitments.
   =2 also uses the mixing tank treatment, but the advanced gas rate is not added to the produced gas rate to form the available gas rate prior to fuel and sales gas commitments. This was the default 99A advanced gas treatment.
31. If > 0 well and group names are not internally converted to upper case.
32. This switch overrides some corrections made to the well calculation for the 99A release, restoring compatibility with earlier releases to allow you to assess the importance of the changes. If:
   = 1 the injection well potential calculation reverts to its pre-99A state, when a THP limit imposed on an injection well was used to set the potential flow rate if the THP value at the BHP limiting potential flows was less than this limit. For the 99A release this calculation has been corrected, with the THP limit now acting as an upper bound for injection wells. This correction only affects runs with injection wells which have THP limits set and are subject to group control, since the default well guide rates are set equal to their potential rates.
   =2 the group and field liquid compositions revert to the pre-99A state, when they were calculated by summing the well component volume rates instead of the well component molar rates. The change affects the reported group and field liquid component mole fractions.
   =10 both the above revert to their pre-99A state.
33. If > 0 the error message associated with calculating a FACTLI value less than 0.9 or greater than 1.1 when using compositional equilibrium option 1 (10th item in EQUIL keyword) is downgraded to a warning, allowing the run to proceed.
34. If > 0 the multisegment well having this well number (wells are numbered in the order they are input) will take two extra steps to help converge when crossflowing. Firstly, the multisegment well primary variables at the start of each well calculation are initialized to the solution at the end of the last timestep. Secondly, during the solution of the flow control rate constraint, the search for the BHP puts extra weight on bisection rather than Newton. This may result in more iterations being taken to converge the well, but it helps convergence in difficult situations. If this item is set to 10000 then all multisegment wells use the above.
35. Not used.
36. This switch overrides some corrections made to the well calculation for the 2000A release, restoring compatibility with earlier releases to allow you to assess the importance of the changes. If
the top-up phase identification calculation reverts to its pre-2000A state, where a phase is assigned as the top-up phase if a group is under a voidage replacement or voidage rate control mode when injecting this phase. The 2000A method assigns a phase as the top-up phase if a group level voidage replacement or voidage rate injection limit or target is associated with this phase.

the simulator reverts to the pre-2000A treatment of completion D-factors entered using COMPDAT or COMPDATL. This treatment applies the D-factors directly to the completions. The 2000A treatment of positive completion D-factors is to take them as applying to the well and scaling them to values appropriate to the completions.

the simulator reverts to the pre-2000A treatment of the end run flag in WECON and GECON. If this flag is set to Y, the run is stopped only on violation of a minimum economic rate limit. The 2000A behavior is to stop the run if the well (or all the producers in the group) have been shut/stopped for any reason.

the simulator reverts to the pre-2000A treatment of well revival as controlled by WTEST. If all the connections of the well are closed, the well will not be revived. The 2000A behavior is to open all connections shut by an automatic workover and compare these connection flows against well and connection economic ratio limits; if any connections now obey these limits, the well is declared open again.

the simulator reverts to the pre-2000A treatment of a well’s bottom hole pressure reference depth that is defaulted in WELLSPEC, WELSPCS or WELSCPL. This reference depth is set equal to the center depth of the grid block containing the first connection defined for this well. The 2000A behavior is to use the center depth of the grid block containing the topmost connection of the well if the bottom hole pressure reference depth is defaulted.

the simulator reverts to the pre-2000A treatment for closing a well that is unable to flow, that is the well is stopped. The 2000A behavior uses item 9 in the WELSPECS keyword to determine whether the well is shut or stopped, with the default action (which applies if the well is specified with the WELSPC keyword) being to shut the well.

the simulator reverts to the pre-2000A treatment for switching a group to a passive control mode because it cannot meet its target without all its subordinate groups and wells first moving onto individual control. The 2000A treatment improves the behavior of the group control facility in handling subordinate wells or groups which are instructed to produce at a higher rate than a group’s target.

the simulator reverts to the pre-2000A treatment of not applying a D-Factor term to the water flow equation for an injecting completion in a crossflowing gas injector. The 2000A treatment applies the D-Factor mobility multiplier to all the injecting phases, which is consistent with the calculation in ECLIPSE 100.

the simulator reverts to the pre-2000A treatment of the injection system, whereby the calculation is performed once per Newton iteration. The 2000A treatment has iterative loops over the injection calculation when advance or make-up gas is present or availability limits have been requested (see keywords WAVAILIM and GCONTOL).

the simulator reverts to the pre-2000A treatment of cases where the source of injection gas is a single separator stage (see GINJGAS item 5) and there are fuel or sales targets that take precedence over injection (see WTAKEGAS). Previously, the fuel and sales gas composition was that of the complete separator, whereas in reality the composition must change to reflect the fact
that an amount of injection gas with the composition of a single stage has been removed. The new treatment allows for this change in composition, by performing the injection calculation first at a rate that is limited by higher priority fuel and sales requirements.

Setting the switch to 100 activates all the above reversions.

37. If > 0 the simulator reverts to the pre-2000A treatment of the **DPCDT** keyword, in which the limiting of the capillary pressure changes occurs after table look-up. The 2000A treatment limits the capillary pressure associated with each phase.

38. If >0 the **DPCDT** limiting applies to both explicit and implicit cells, which was the pre-2000A treatment. The 2000A default is to only have the **DPCDT** limiting on explicit cells.

39. Reserved for internal use. Do not use this.

40. Enables screening of the oil, water and gas saturations from a **GETSOL** (Thermal option).

41. If > 0 this disables the 2000A treatment of hydrocarbon-water relative permeabilities, where the endpoints of the gas and oil-water curves are scaled before interpolation based on the Li correlation (see "Near critical oil and gas relative permeabilities" in the *ECLIPSE Technical Description*).

42. This switch controls options that may aid convergence in dual porosity cases. It is possible to reach a position where one phase’s potential gradient goes from matrix to fracture and another phase’s goes from fracture to matrix. This becomes a problem if the upwind mobilities of the respective phases are small or zero. Typically these situations occur rarely, but they can cause severe convergence difficulties. If:

   =1 invokes an option to set the matrix pressure change over a Newton iteration to the fracture value.

   =2 the transmissibility between matrix and fracture is lowered for a single timestep to allow convergence. Note that in principle this slightly modifies the resulting solution, but this modification is unlikely to be significant in most field examples.

43. This switch overrides some corrections made to the well calculation for the 2001A release, restoring compatibility with earlier releases to allow you to assess the importance of the changes. If:

   =1 the defaulting for Kh (permeability x thickness) in the **COMPDAT** keyword reverts to its pre-2001A state, where an unset Kh value is calculated from the grid block data regardless of whether the connection transmissibility factor is set. The 2001A method calculates an unset Kh value from a given connection transmissibility factor and geometric denominator term calculated from grid block data. Furthermore in 2001A, if both Kh and the connection transmissibility factor have been set, the geometric denominator term is calculated from these two values.

   =2 no warning or error messages are generated when a well completed in an LGR is specified in keywords that require connection values and are not designed for LGR wells. The connection values are assumed to refer to the simulation grid with the LGR positioned below the global grid. The 2001A behavior is to generate an error message in this situation unless the connection values are all defaulted and the action of the keyword thus applies to every connection in the well. From 2010.1 onwards this error message has been replaced by a warning message and the offending well is skipped.

   =3 groups that were set under the control of a higher level group (‘ **GR** ’) with keywords **GRUPPROD** or **GRUPINJE**, but did not have a guide rate explicitly set (using **GRUPGR**), are not reset to a passive state and are reported as under ‘ **GR** ’ control. This reported mode does not accurately reflect the way the group is solved by ECLIPSE 300. The 2001A behavior is to report a group that has been set under ‘ **GR** ’ control without a user set guide rate as a passive group. This switch
also reverts ECLIPSE 300 to assigning the default group guide rate to group’s potential. The default value for group guide rates in 2001A is zero (which means ‘undefined’), and groups only have guide rates if they are set explicitly. This is now consistent with ECLIPSE 100.

Setting the switch to 100 activates all the above reversions.

44. This switch may be used to control the calculation of explicit hydrostatic heads, either segmented (see item 12 of WELSPECS keywords) or averaged (see item 1 of this keyword).

The explicit head is calculated at the start of each timestep and remains constant over the timestep.

- If set > 0, the head is also averaged with its value of the previous timestep, to damp out any oscillations that may be caused by the use of explicit quantities. The averaging depends on the value n of this switch: \( H(\text{used}) = (H(\text{current}) + nH(\text{previous})) / (n + 1) \)
- If set=0, the head is averaged as above with n=2
- If set<0, no averaging takes place.

45. This switch may be used to control various aspects of how wells are solved under THP control.

In the 2001A release of ECLIPSE 300 a predictor has been incorporated that estimates the solution for a THP controlled well by finding the intersection between the latest linear inflow performance relationship (IPR) for the well and the cross section of the VFP curve at the target THP and the well’s latest water and gas fractions. This prediction can often speed up the full solution of the well.

In addition to introducing a predictor, a new scanning mechanism was introduced in the 2001A release. A scan is called when the THP control algorithm struggles to find a solution (this is usually when the well is near to the point of being unable to flow under THP control). A successful scan supplies two BHP values that result in two THP values that straddle the target THP; this then makes the full solution of the well much easier. The old scanning mechanism attempted to find straddling THP values by numerous well solutions over a spread of BHP values. This technique can be expensive, especially for the Multisegment Well Model. The new scan only solves the well at flow values that correspond to those in the VFP table. This accurately models the point at which the well is unable to flow without resorting to too many well solutions.

You may switch the predictor on and off and switch between the old and new scanning techniques as follows, if:

=1 the predictor is not used, but the new scanning technique will be used if necessary. This may be used if the prediction appears unreliable.

=2 the predictor is used, but no scan is called should the THP algorithm struggle to find a solution. This should be quicker than the default case, with the decision to close a well (because it is unable to flow on THP control) being based on the predictor.

=3 the predictor is used and if necessary both new and old scanning mechanisms are used. This should slower than the default case, but the well may flow a little longer under THP control before closing (if the timestepping resolution allows this to be seen).

=4 the predictor is not used and the old scanning mechanism used if necessary. This then reverts the THP control algorithm to the 2000A state.

46. A positive value sets the number of Newton iterations within a timestep in which a well can be revived after being closed because it is unable to flow given the current reservoir conditions. The default behavior for the 2001A release when not using the Thermal option is to revive such a well only during the first NUPCOL Newton iterations. When the Thermal option is used, the default is to allow 21
revivals within a timestep before the well is closed. If this switch is set to a negative value, no well revivals are permitted. If however the switch is set to -999 the simulator reverts to the pre-2001A treatment, whereby the number of revivals permitted within a timestep was fixed at 21.

47. Number of network balancing iterations in which a well can be revived after being closed because it cannot operate against its current THP.

If set = 0, this defaults to max (4, NETBAL/2) where NETBAL is the maximum number of network balancing iterations allowed (see the NETBALAN keyword).

48. This option will turn off either the gravity or the porosity-stress relation in the rock stress calculation of the ECLIPSE 300 Geomechanics option. If:

=4 the gravity term is inactive in the stress equations

=6 the porosity-stress relation is inactive and porosity remains constant

=10 this is equivalent to both of the above.

49. Activates prototype group control logic that may help in problematic runs when wells and/or groups have significant flows that are unavailable for higher group control.

50. Reserved for use by FrontSim.

51. Governs the frequency at which the well potentials are recalculated during network coupling iterations. The potentials are used to calculate well and group guide rates, which govern the apportioning of group flow targets unless they are to be handled by automatic chokes. If:

=0 this defaults to recalculating the potentials at every network coupling iteration. The potentials will thus remain consistent with the latest THP limits from the balanced network.

=1 the potentials are calculated only once in each of the first NUPCOL Newton iterations of the timestep, before each set of network coupling iterations begin. This is the same as the frequency for recalculating the well potentials when there is no network; however, when a network is present the potentials are not fully consistent with the latest balanced THP limits.

52. In 2001A and later releases, the horizontal thermal transmissibility is based on the full cell cross-sectional area and does not include a net-to-gross factor. If this option is set to 1, the pre-2001A calculation, which includes net-to-gross, is used.

53. Activates a simple algorithm, using direct table lookup values, for oil relative permeabilities that are used when calculating the blocking factor in the Generalized Pseudo Pressure option (see the PSEUPRES keyword). By default in 2001A and later releases, the oil relative permeabilities used in the blocking factor calculation are found from a full three-phase calculation in the same manner as grid cell permeabilities. Should this more accurate calculation prove prohibitively expensive, the simpler method may be used by setting this option to 1. (This uses an internal version of SOF3 for $k_{rog}$ and $k_{row}$ values that are combined using the default model for three-phase oil relative permeability as described in "Saturation Functions" in the ECLIPSE Technical Description.)

54. This switch overrides some corrections made to the well calculation for the 2002A release, restoring compatibility with earlier releases to allow you to assess the importance of the changes. If

=1 for wells connected to the innermost grid block of a radial grid, the connection transmissibility factor is calculated using the innermost grid block radius. By default, the wellbore radius is used in this calculation.
=2  for wells with THP limits set, in some circumstances, the potential rates will be calculated at the well’s BHP limit even though the rates at the THP limit are lower and thus should be assigned as the well’s actual potential. By default, the well potentials are correctly calculated.

55. Not used at present.

56. If =1, then in black oil mode, the oil formation volume factor and the viscosities are interpolated in 1/bo and 1/(bo.Vo) as in ECLIPSE.

If =2, and using PVDG, the gas formation volume factor and viscosities are also interpolated in 1/bg and 1/(bg.Vg). This option has no effect on PVTG, where the formation volume factor is interpolated in 1/bg but the viscosity is interpolated directly.

57. If =1, then the pre-2002A equilibration behavior is restored.

If =2, then the pre-2012.1 equilibration behavior is restored: if a saturation pressure calculation fails the results are still used to determine the vapor composition below the gas-oil contact or the oil composition above the gas-oil contact.

58. Controls the time truncation error (tte) based timestep selection:

>0 then the timestep is based on the \( \sqrt{\text{target/tte}} \) rather than \( \text{target/tte} \).

<3 then the calculation is based on (normalized) mass change.

=3 then the calculation is based on the saturation change

>3 then the calculation is based on both masses and saturation.

59. If set >0, in runs using flux boundary conditions certain error conditions are reduced to warnings to allow the run to proceed.

If set >0, in a full field DUMPFLUX run, any error messages about wells connecting to multiple flux regions are replaced by warnings. These remain strict errors in the reduced flux USEFLUX run when wells connect both inside and outside the USEFLUX region. This switch is useful in cases with many flux regions where it is known a priori which flux region numbers constitute the USEFLUX region (using FLUXREG in the reduced run) and that there is no problem with the placing of wells in the reduced run.

60. This option can be used with the SKIPSTAB keyword. The simulator performs a rigorous one-phase stability check on every \( n \)th non-linear iteration. Therefore a value of 1 switches the SKIPSTAB option off and a value of 0 switches it back on again.

61. Restores pre-2002A convergence behavior of the linear solver.

62. This option must be set in the RUNSPEC section. If set >0, all wells declared throughout the simulation that are not multisegment wells are internally converted to single segment multisegment wells. The Multisegment Well Model is thus used in preference to the standard well model; this has been found to improve numerical convergence, and thus reduce timestep chopping, in some simulations.

63. If set = 1, this prevents extrapolation of the VFP tables in THP, WFR, and GFR during the calculation of the branch pressure drop in the Network Option. If a value lies outside the range of the VFP table, the branch pressure drop is calculated at the nearest tabulated value. Otherwise, the VFP table is extrapolated linearly in these quantities if they are outside the range of the table.
64. Reserved for Geomechanics

65. This switch may be used to change the type of density calculation for the wellbore hydrostatic head (see item 12 of the WELSPECS keyword) as follows, if:

   \( = 1 \) the default method is set to ‘AVG’. This was the default prior to the 2002A release.

   \( = 2 \) all wells have their wellbore density calculation method changed to ‘SEG’. This switch applies to all currently defined wells and all wells to be defined in the remainder of the simulation, and overrides item 12 in the WELSPECS keyword.

   This override switch would be useful when running from a pre-2002A flexible restart in which all the wells, by default, use the ‘AVG’ density calculation method.

   \( = 3 \) all wells have their wellbore density calculation method changed to ‘AVG’. This switch applies to all currently defined wells and all wells to be defined in the remainder of the simulation, and overrides item 12 in the WELSPECS keyword.

66. If set > 0 this switch reverts to the 2001A linear solver. From 2002A onwards the default linear solver for serial and parallel runs is WARP. The use of this linear solver does not affect the results, only the performance, which normally improves.

67. This switch may be used to change the pressure convergence tolerance in the well model. When solving a well under flow rate or THP control, the well’s BHP is adjusted such that the flow rate or THP of the well approaches the target value. The well is deemed to be solved when changes in its BHP are less than this pressure tolerance.

   The default pressure tolerance is divided by ten to the power of the value set in this switch. So setting this switch to 2 tightens the convergence tolerance by a factor of one hundred. Setting a negative value slackens this convergence tolerance.

   Tightening this tolerance may be necessary if flow rate targets are not being accurately met or if the target is small enough to result in well drawdowns that are similar to the default pressure tolerance (0.01 psia).

68. If set > 0, this switch turns on non-linear speed-up due to Gain.

   Note that this is also affected by the setting of Item 19. Both items must be set to ‘on’ for the non-linear speed-up due to Gain to be used (so, by default, Gain is generally ‘off’).

69. If set > 0 this switch restores the 2001A tunings, that is restores changes made to default values of the TSCRIT and AIMCON keywords. The 2002A setting are equivalent to having the AIM1 argument set in the TOPT keyword in the 2001A release. The changes made are

   - 11th item in keyword TSCRIT has changed, this limits the timestep size after well modifications. For AIM runs this is now set to 40, thus allowing one-month timesteps, which is often desirable and achievable in history-matching data sets.

   - The default procedure for selection of implicit cells during an AIM run has changed, that is, the default setting of the AIMCON keyword has changed. The new setting often chooses fewer implicit cells for the early time in a history-matching case (water-injection), and more implicit cells during a gas-injection prediction phase.

   These settings are believed to produce better answers more efficiently.

70. This switch changes the behavior of the ACTNUM keyword in dual porosity cases. By default inactivating a cell in either matrix or fracture has the effect of inactivating both cells. Setting this
switch to >0 means that only the cell specified is inactivated. This was the default behavior prior to the 2002A release.

One exception to the new behavior is when a fracture cell contains a numerical aquifer (see keyword AQUNUM). In that case, inactivating the corresponding matrix cell does not automatically inactivate the fracture cell with the aquifer inside.

71. Reserved for Geomechanics

72. Base IMPSAT timesteps on the average throughput rather than the maximum.

73. Use throughput when selecting implicit cells in IMPSAT cases. If
   =1 recalculates the vector suns only (null check)
   =2 uses the throughput to set implicit cells.

74. Uses a more rigorous calculation of the saturation pressure. This calculation is significant slower than the default calculation, but gives more accurate answers for fluids near the critical point.

75. By default, all solution data requested under the RPTSOL keyword will be output for any local grid refinements present. Setting this switch to > 0 implies that the local grid output will not be output for the SOLUTION section.

76. If set >0 this switch reverts to the 2002A treatment of combined hysteresis and miscible flooding for the calculation of relative permeabilities. In this previous treatment the critical saturation for the immiscible part was based on the drainage curve rather than on a scanning curve if relevant. Also the critical point of the miscibility curve was assumed to be zero, which may not be the case if the MISCNUM keyword is used.

For models with the relative permeability hysteresis option (see HYKR keyword) or the MISCNUM option, setting this value >0 also reverts to pre-2005A behavior for the calculation of the effect of the surface tension on the relative permeability scaling end points.

77. Requests output of the computed gradients of all optimization constraints specified in keyword OPTCONS, while skipping the actual Reservoir Optimization process. The gradients are stored in files (see keyword SOPG in "Optimization State Files" in the ECLIPSE File Formats Reference Manual) available to any optimization software running outside of ECLIPSE.

78. Resets the PVT region number for all wells. In runs with more than one PVT region (see item 2 of the TABDIMS keyword), the 2004A simulator sets a PVT region number for each well, according to the PVT region number of the completion cells. This option allows the user to override this PVT region number, and set a new PVT region number for all wells. Prior to 2004A the properties of fluids in the wells were calculated using a PVT region number of 1. Therefore setting this item to 1 will give the pre-2004A behavior.

79. This switch overrides some corrections and improvements made to the well calculation for the 2004A release, restoring compatibility with earlier releases to allow assessment of the importance of the changes. If:
   =1 when a group becomes passive, wells and groups at a lower level will become subject to rate control from groups at a higher level even if the passive group was set ‘not free to respond to a higher level production rate target’ (see item 8 in GCONPROD or GCONINJE). The default behavior in 2004A is that the ‘not available for higher level rate control’ restriction remains in effect even if the group to which it is applied becomes passive.
   =2 when performing the top-up phase injection calculations for a group, any group flows in the non-top-up phase are treated as unavailable for group control flows (that is in the same manner as

Keywords

1558
satellite group flows). The default behavior in 2004A is to follow the ECLIPSE 100 methodology and directly reduce the group’s voidage target by the non-top-up phase flows - this is a simpler and more robust approach.

=3 when summing over wells to form a group’s potential flow rates, well efficiency factors are not taken into account. The default behavior in 2004A is to multiply each well’s potential rate by its efficiency factor when forming group values.

Setting the switch to 100 activates all the above reversions.

80. Reserved for ComSim
81. Reserved for ComSim
82. This switch allows you to change the way in which Forchheimer effects are incorporated in the Generalized Pseudo-Pressure calculation for a well when item 4 of PICOND is set to YES. If:

=1 the Forchheimer flow multiplier is calculated using a B-parameter equation based on the fluid properties of the connected cell (see Non-Darcy flow in the ECLIPSE Technical Description). By default the B-parameter is calculated using the fluid properties and saturations at each pressure point in the pseudo-pressure integral.

=2 Forchheimer effects in the Generalized Pseudo-Pressure calculation are switched off and only Capillary Number velocity-dependent relative permeability effects are incorporated in this calculation. This was the behavior of pre-2004A releases.

83. Post 2007.1 the Multisegment Well Model uses the modified solution update method by default. The modified method can improve convergence for difficult problems, but may be slower in more straightforward cases. If:

=1 the pre-2007.1 multisegment well update method is restored.

84. Number of times a single timestep can be consecutively chopped because a well has switched from rate control to pressure control and died in the same timestep. If:

=0 defaults to 1.

= negative prevents timestep being chopped for this reason.

85. If set to 1, use the high mobility injection well model for all injectors. See WELSPECS item 16.
86. If set to 1, revert to pre-2004A weighting of average phase potentials by hydrocarbon volume rather than phase saturation. The calculations of phase potentials are detailed in “Potential Calculations” in the ECLIPSE Technical Description.
87. If set to 1, revert to pre-2004A behavior if JFUNC and PCW or PGW are present in the dataset.
88. This switch overrides some corrections and improvements made to the well calculation for the 2005A release, restoring compatibility with earlier releases to allow assessment of the importance of the changes. If:

=1 connection voidage rates are calculated at the reference pressure for the well (see item 13 of WELSPECS). The default behavior in 2005A is that a connection’s voidage rate is calculated at the reference pressure for the fluid-in-place region of the connected grid block.

=2 derivatives of the voidage control well residual are calculated in the pre-2005A manner.
=3  a number of improvements to the segmented wellbore pressure head calculation for crossflowing injections wells are disabled.

Setting the switch to 100 activates all the above reversions

89. If set to 1, enable the Finite Difference stress calculation to run with corner-point geometry.
90. If set to 1, enable SAMG solver convergence statistics to be printed in the debug file.
91. If set to 1, revert to pre-2005A linear solver control. This is not recommended.
92. Alternative timestepping strategy will be implemented.

In thermal runs (2006.1 onward) this option is automatically selected and can be turned off by setting a positive value

93. If set to 1, do not scale by the fracture gas saturation in the sorption rate equation for a Coal Bed Methane Model.
94. If set to 1, revert back to the pre-2006.1 well name and well list template matching.
95. This switch controls how the gas fuel and sales will be scaled back if the sum of the fuel, sales and reinjection exceeds the amount of gas available. If:

=0 (default) the sales and fuel are both scaled back by the same amount. If WAVAILIM is not used then no scaling back takes place.

=1  the priority order specified in WTAKEGAS is used to decide which of the fuel and sales to scale back first.

=2  there is no scaling back of the fuel and sales.

=3  the sales and fuel are both scaled back by the same amount whether WAVAILIM is not used or not.

=4  the priority order specified in WTAKEGAS is used to decide which of the fuel and sales to scale back first. If WAVAILIM is not used, then no scaling back takes place.

96. This switch controls some options related to the calculation of the velocity dependent skin factor (see "Forchheimer models" in the ECLIPSE Technical Description for more information). If:

=0 (default), the pressure equivalent radius term is not included (see “Forchheimer parameter based well D-factors” in the ECLIPSE Technical Description is used), and the new unit-dependent constants are used.

=1  the pressure equivalent radius term is included (see “Forchheimer parameter based well D-factors” in the ECLIPSE Technical Description is used), and the new unit-dependent constants are used.

=2  the pressure equivalent radius term is not included (see “Forchheimer parameter based well D-factors” in the ECLIPSE Technical Description is used), and the pre-2006.1 unit-dependent constants are used. Use this value to revert to the pre-2006.1 behavior.

=3  the pressure equivalent radius term is included (see “Forchheimer parameter based well D-factors” in the ECLIPSE Technical Description is used), and the pre-2006.1 unit-dependent constants are used.
97. This switch limits the timestep taken by the simulator after a multisegment well changes control mode. If set to 1 the pre-2006.1 limit of 5 days is applied, otherwise no limit is enforced.

98. Relax the monotonicity restriction of the ROCKTAB keyword; warnings are suppressed for non-monotonic pore volume and transmissibility multiplier values.

99. If set to non-zero, and the original CN model for velocity dependent relative permeabilities is being used, then the behavior returns to the pre-2006.1 use of velocity dependent relative permeability calculations in the generalized pseudo-pressure calculations. Pre-2006.1, the velocity dependent correction to the relative permeability was performed after the iterations to determine the generalized pseudo-pressure. At 2006.1 the velocity dependent correction to the relative permeability is performed within the iterations to determine the generalized pseudo-pressure. This switch also reverses some bug fixes in the original CN model for back compatibility with pre-2006.1 results.

If set non-zero, and the alternative CN model for velocity dependent relative permeabilities is being used, then the pre-2007.1 behavior is used. See "Alternative model for capillary number effects" in the ECLIPSE Technical Description for a description of the pre- and post-2007.1 behavior.

100. Reserved for future development.

101. If set to 1, then the test for zero thickness blocks in the X- and Y-direction is removed. Doing this can improve the pore volume calculation for nasty grids, for example, XY in map feet; IJ orientation with I up the page, J across.

102. Reserved for future use.

103. If this switch is non-zero, the connate gas saturation is allowed to be non-zero. This can be used in cases where non-zero connate gas is required and also provides back compatibility with pre-2004A behavior for dual porosity equilibration.

104. If this switch is non-zero, then the PBGS surface volume production balancing group control mode balances the produced surface gas volume, instead of the excess surface gas volume, with the specified fraction (or multiple) of the injected surface gas volume.

105. Restores pre-2006.2 behavior regarding tilted aquifer equilibration and calculation of the block equilibration pressures. If:

   > 0 pre-2006.2 behavior is restored with the equilibration tilt correction not being applied to numerical aquifers and the equilibration block pressure taken as an average pressure over the sampling points specified by item 9 of the EQUIL keyword.

   =0 tilt correction is applied to numerical aquifers if item 9 of the EQUIL keyword is positive and the block equilibration pressure is taken as the central pressure.

   <0 tilt correction is applied to numerical aquifers if item 9 of the EQUIL keyword is positive but the block equilibration pressure is taken as the average pressure.

106. If set above 5000, this increases the maximum number of internal allocatable arrays to the value specified. This option is intended for use in special circumstances, for example when there are a large number of equilibration regions and the previously hard-wired limit of 5000 is insufficient. A message will be output by the simulator indicating when you should use this option.

107. Restores pre-2007.1 behavior of the drift flux model for small flows and hold-ups. If:

   > 0 the 2006.2 behavior is restored, this affects the drift flux calculations in the presence of small values of flow rate and hold-up.
> 1 the pre-2006.2 behavior is restored, this only affects the reporting of segment phase volume flow rates and velocities for small hold-ups

108. By default, gas injected during the surface gas production balancing group control mode is reported as reinjected gas, and the reported excess and export gas is based on this injected gas (not just gas formally reinjected during a reinjection group control mode). If this switch is non-zero, then this production balancing injected gas is not treated as reinjected gas. This affects the reporting of reinjected, excess and export gas in the gas accounting tables in the PRT file, and also in the values of these quantities written to the summary files.

109. For developer use.

110. This switch can used to revert to 2006.1 behavior. It is used internally to correct the saturations if the oil saturation becomes negative. In that case the gas-water capillary pressure is used, and the new behavior handle situations where the gas-water capillary pressure becomes negative.

111. In black oil simulations, with both dissolved gas and vaporized oil, the condition \(1 - R_{ssat} \times R_{vsat} < 0\) should hold. If it does not hold, the black oil flash may not be able to determine whether a single phase fluid is oil or gas. If set to 1, the pre-2007.1 behavior is used where single phase fluids default to oil. If set to 2, single phase fluids default to gas.

112. This option must be set in the RUNSPEC section. If set to a non-zero value, this reverts to the pre-2007.1 default behavior of reporting mass rates and totals in long tons (instead of short tons).

113. If set to various non-zero values, this restores aspects of the pre-2007.1 behavior regarding injection of gas and mixtures.

- If divisible by 2, use the pre-2007.1 calculation of availability of items in mixtures specified using WINJORD.
- If divisible by 3, use availability fractions when calculating mixture requirements. (These availability fractions are not used from 2007.1, instead a new calculation of the availability is performed.)
- If divisible by 5, use the pre-2007.1 calculation of mixture potentials when calculating the availability fractions for mixture requirements. (Bugs were fixed in this calculation in 2007.1 to prevent crashes.)
- If divisible by 7, use the pre-2007.1 treatment of gas specified using the injected fluid nature ‘ GAS ’ in the GINJGAS, GRUPINJE, WELLINJE, WINJGAS, WINJMIX and WINJORD keywords. From 2007.1 these are internally converted to injection fluid nature ‘ GV ’ with its source being the field separator (which has better gas accounting and r-injection behavior).
- If divisible by 11, use the pre-2007.1 treatment of reinjection for mixture items from a separator stage. Before 2007.1, these mixture items were effectively ignored in complex gas accounting (that is, where there is fuel and/or sales gas, with intermediate separator stage gas reinjection).
- If divisible by 13, complex gas accounting (that is, where there is fuel and/or sales gas, with intermediate separator stage gas reinjection) uses the pre-2007.1 treatment of reinjection in calculations of excess and export gas.
- If divisible by 17, gas from a satellite group is not reported as separator stage 1 gas for the group.

114. If set to a non-zero value, a simpler form of the max NaCl solubility in water is used for the CO2STORE option.

115. If set to 1, the default values of the water formation volume factor of the solubility table used in the CO2SOL option are calculated using the pre-2007.1 method. This affects the water density calculation.
116. If set to a non-zero value the pre-2007.1 implementation of the BRINE model is restored. This affects the equilibration of the initial solution.

117. If set to a nonzero value the NTG values will also apply to the calculation of the rock volumes. This is the pre-2007.1 behavior.

118. If set to a non-zero value various aspects of the pre-2007.1 ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS, ACTIONX) implementation are restored. The default behavior from 2007.1 onwards is that ACTION (ACTIONR, ACTIONX) conditions involving regions are evaluated using the summary quantities from the current timestep, the maximum timestep length after a well modification is respected, and the summary data is output before processing the ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS, ACTIONX) conditions. If set to:

-1 the pre-2007.1 timestepping sequence can be recovered. Specifically, the 2007.1 order of evaluating the ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS, ACTIONX) conditions and the output of the summary data is followed, but the maximum timestep length after a well modification is not imposed.

119. If set to 1, the water properties of water analytic aquifers are calculated using the PVT tables for the GASWAT, CO2STORE, CO2SOL or COMPW options.

- For GASWAT or CO2STORE, this was the pre-2007.1 method. By default, the water properties are currently calculated using the EoS.
- For the multi-component water option (COMPW), this was the pre-2008.1 method. By default, the water properties are currently calculated using the water components data.
- For the CO2SOL option, this was the pre-2008.1 method. By default, the water properties are currently calculated using the solubility table.

120. If set to a positive number, the pre-2006.2 handling of a dual porosity system is restored. Otherwise, active matrix cells that are connected to inactive fracture cells (because of a MINPV criterion) are inactivated.

121. If set to a positive number, that number is taken as an upper limit for the blocking factor calculated by the generalized pseudo-pressure special inflow equation. If a zero (that is default), or negative value is specified, then no limit is applied. This can be used to prevent convergence problems when blocking factors greater than unity are generated. See "Generalized pseudo-pressure in ECLIPSE 300" in ECLIPSE Technical Description for more information.

122. Generates warning messages from the AQUANCON keyword when cells cannot be connected to the aquifer because there is an active cell on the aquifer side of the current cell.

123. If a zero (that is default), or negative value is specified, then no normalization is applied. If set to 1, then normalization of the track tracers concentration is applied for each well and the calculation of injected track tracers concentration is also performed. This can be used to prevent discrepancy
problems between field tracers’ characteristics and field characteristics. The injected track tracers’
concentration can be monitored in a similar way as ordinary track tracers. The naming of the injected
tracers is done automatically in a similar manner (for example, A11, B12). If the ‘REGONLY’ option
is used then the tracer A11 refers to region A and injected component 1, B12 to region B and injected
component 2, and so on. For example, mnemonic FTTLB12 can be used in the summary section to
monitor FTTL for the injected tracer B12.

124. If set to a positive number, a better hysteresis model for water capillary pressure is activated.

125. If set to a positive number the phase stability check is modified slightly in order to prevent the H2O-
rich phase, in a GASWAT run, being labeled as gas (default from 2008.1). The pre-2008.1 behavior is
achieved by putting the value to -1.

126. If set to a non-zero value, the pre-2007.1 behavior when the INTPC option for integrated capillary
pressure is used with the alternative gravity drainage model specified by GRAVDRM is restored.
Although INTPC was previously allowed with GRAVDRM, the pre-2007.1 behavior did not lead to the
correct final recovery in this case, whereas the 2007.1 behavior should do so.

127. If set to a non-zero value the block-by-block inter-region flowing fractions calculation for the table
GPTABLEN look up is activated. By default, the inter-region flowing fractions are calculated using the
region-averaged values.

128. If set to a non-zero value the pre-2007.2 tolerances used to calculate the derivatives of immobile fluids
are restored. This can lead to large elements in the Jacobian and affect the convergence of the solution.

129. If set to a non-zero value, the pre-2008.1 behavior for scale deposition is set. Scale deposition will
only be applied to producing completions (which could belong to producers or crossflowing injectors).

130. This option controls three features of the gas accounting used when there is reinjection of a fluid from
a separator stage, either as part of a mixture or alone.

• If set to 1,3,5 or 7 the pre-2004A behavior for gas accounting is used when there is reinjection of
a mixture fluid (that is, one defined using the WINJMIX or WINJORD keywords) in which a
mixture item comes from a separator stage.

• If set to 2, 3, 6 or 7 the pre-2007.2 behavior for gas accounting is used when there is reinjection
of separator stage fluid (defined using GINJGAS), with sales and/or fuel targets (defined using
GRUPSALE or GRUPFUEL) not being respected appropriately in some circumstances.

• If set to 4, 5, 6 or 7 the pre-2007.2 behavior for sales and fuel target checks is restored, with
tighter checks on these targets in place in most circumstances.

131. This option controls the composition of water in analytic aquifers.

• For the GASWAT or CO2STORE options, if set to 1, the pre-2007.2 behavior for water analytic
aquifers is used where the water leaving / entering the reservoir is made up entirely of the “H2O”
component.

• For the multi-component water option (COMPW), if set to 1, the pre-2008.1 behavior is used
where the water leaving / entering the reservoir is made up entirely of the last water component.

• For the CO2SOL option, if set to 1, the pre-2008.1 behavior is used where the water leaving /
entering the reservoir is made up entirely of the “WATER” component.

132. This option controls a number of changes to the generalized pseudo-pressure calculation.

• If set to a non-zero value divisible by 2 the pre-2007.2 behavior for generalized pseudo-pressure
is applied. This restores minor bugs in the evaluation of the mobility integral.
- If set to a non-zero value divisible by 3 the mobility at drainage radius used in the denominator of the blocking factor calculation is calculated in the same way as the integral mobility. The default behavior is to take the bulk cell mobility. This may be useful for some cases, but can lead to instabilities in the blocking factor calculation.

- If set to a non-zero value divisible by 5 the pre-2008.1 behavior for generalized pseudo-pressure is applied. The mobility calculated for the blocking factor integral at pressures above drainage radius pressure and below completion pressure include velocity dependent relative permeability effects at these limiting pressures - before 2008.1 these effects were ignored. This only affects the calculation if capillary number model 1 is used: see \texttt{VELDEP}.

- If set to a non-zero value divisible by 7 the pre-2008.1 behavior for velocity dependent relative permeability in the generalized pseudo-pressure calculation is applied for radial grids. This only affects the calculation if capillary number model 1 is used: see \texttt{VELDEP}.

133. If set to 1, the residual in the WARP linear solver is calculated with greater numerical accuracy.

134. If set to 1, the flows output by the \texttt{RPTRST}, \texttt{RPTSCHED} and \texttt{RPTSOL} keywords are calculated at the start of the timestep rather than the end. This was the pre-2008.1 behavior.

135. If set to 1, an alternative implementation for predicting the energy density in the \texttt{HEATER} algorithm is activated.

136. If set to a non-zero value, apply the restrictive well-reservoir convergence check.

137. If set to a positive value, do an extra flash at the end of an accepted timestep (updates the fluid properties for reporting).

If set to a negative value, do not perform this extra flash in \texttt{TRACER} runs (for pre-2008.1 back compatibility).

138. If set to a non-zero value, this reverts to the pre-2008.1 method when determining the light and heavy components for the saturation chop calculation (equation of state mode).

139. This option alters the behavior of gas accountancy, reverting various elements to pre-2008.1 behavior.

- If non-zero and divisible by 2 the pre-2008.1 calculation of reinjection fraction method is used for separator stage reinjection.

- If non-zero and divisible by 3 the pre-2008.1 calculation of advance gas amount with \texttt{WINJORD} or \texttt{WINJMIX} is used. This causes advanced gas to be applied for each separator stage used instead of once for the source group or well. It also changes the calculation of injection composition with \texttt{WINJMIX} and the priority of advance gas injection.

- If non-zero and divisible by 5 the pre-2008.1 behavior of basic gas accountancy is recovered. This changes the calculation of sales, fuel and reinjection amounts if sales or fuel gas is specified for groups beneath a reinjection source group in the group tree.

- If non-zero and divisible by 7 the pre-2008.1 behavior of partial gas reinjection with sales or fuel gas is recovered. This changes the calculation of injection rate target to the reinjection fraction applied to available gas before sales and fuel deductions and before advance gas additions. This only applies to the reinjection of gas mixtures defined using \texttt{WINJORD} or \texttt{WINJMIX}.

- If non-zero and divisible by 11 the pre-2008.1 behavior of \texttt{QDRILL} with sales or fuel gas is recovered. This changes the point at which new gas injection wells are drilled. This only applies to the reinjection of gas mixtures defined using \texttt{WINJORD} or \texttt{WINJMIX}.

140. If set to a non-zero value the pre-2007.2 calculation for the lumped tracer connection phase flow, which does not include a Li correlation factor in the wellbore flash, will be used.
141. If set to a nonzero value the segmented well head calculation in BLACKOIL mode reverts to pre-2007.2 behavior. This uses fluid densities for PVT region 1 irrespective of the well PVT region number.

142. This element controls the specification of bottom hole depth. If set to:

1. the bottom hole depth of a well is calculated using the pre-2008.1 algorithm if WELSPECS element 5 is defaulted. This sets the bottom hole depth to the depth of the center of the vertically highest completed block.

2. the bottom hole depth of a well is calculated using the 2008.1 algorithm if WELSPECS element 5 is defaulted. This may not give the expected result in some circumstances.

143. When modeling oil-wet systems with the end point scaling option activated, it is advantageous to be able to scale the water-oil capillary pressure at the maximum water saturation (using keyword PCW) rather than at the entry (connate) water saturation (where the water saturation equals SWLPC). This switch caters for this situation. Of rather less interest, but for completeness, it is possible to scale the gas-oil capillary pressure (keyword PCG) at connate rather than maximum gas saturation using this option switch. If OPTIONS3(143) is non-zero and divisible by 2 then the water-oil capillary pressure is scaled at the maximum water saturation; and if OPTIONS3(143) is non-zero and divisible by 3 then the gas-oil capillary pressure is scaled at the connate gas saturation. If OPTIONS3(143) is set to 1, then any water capillary pressure curves for which $P_{cow}$ is entirely positive are chosen as water-wet and scaled at the entry saturation SWLPC while any for which $P_{cow}$ is entirely negative are scaled at the maximum saturation SWU. For mixed wettability curves, it would be a bit of an illusion to arbitrarily choose which end point to scale at, so setting this option switch to 1 does nothing for such curves. When the SWATINIT keyword is used to match an initial water distribution then the scaling on $P_{cow}$ is more sensible if OPTIONS3(143)=1 when the rocks are either water wet or oil wet (but no mixed wettability rocks). When the SWATINIT keyword is used to match an initial water distribution for mixed wettability rocks and positive capillary pressure dominated then the scaling on $P_{cow}$ is more sensible if OPTIONS3(143)=5 (minimum capillary pressure is fixed). When the SWATINIT keyword is used to match an initial water distribution for mixed wettability rocks and negative capillary pressure dominated then the scaling on $P_{cow}$ is more sensible if OPTIONS3(143)=7 (maximum capillary pressure is fixed). Note that OPTIONS3(143)=5 and OPTIONS3(143)=7 are not scaling the capillary pressure above GOC, therefore, SWATINIT cannot be honored above GOC.

144. If set to 1, any errors resulting from checking the Z-factors are downgraded to warnings.

145. If set to 1, the pre-2008.1 handling of capillary pressure in analytic aquifers will be used.

146. Switch to activate one-phase selection method.

0  Li-criteria + minimum Gibbs free energy. (Default)

1  Preferred phase is LIQUID.

2  Preferred phase is Vapor.

3  Minimum Gibbs free energy and Wilson K-values

4  Minimum Gibbs free energy and Wilson K-values for pressures below 10 atma, otherwise Li-criteria

5  Same as 0
6 Li- criteria (Pre-2008.1 default)

147. Reserved for developer use.
148. If set to 1, the pre-2008.1 handling of the SWATINIT calculation will be used.
149. If set to 1, the pre-2008.1 calculation of pressure equivalent radius for grid blocks at the center of radial grids is used.
150. If set to 1, the pre-2008.1 handling of WTEST after GECON shuts-in a group of production wells is used.
151. If set to 1, only the gas phase of a GASWAT or CO2STORE run is considered as hydrocarbon. This affects the calculation of the summary keywords FPR, FFRH and FPFO, as well as the fluid-in-place reports. By default, the water and gas phases are considered as hydrocarbon.
152. If set to 1, a minimum timestep that fails to converge is re-solved so that the solution accepted is the closest rather than just the last Newton iteration. This option will require more Newton iterations but can help a simulation through a very poor period of convergence.
153. From 2008.1, an improved vapor versus liquid density check is performed. If this item is set to a non-zero value, the pre-2008.1 behavior will be restored.
154. If set to a non-zero value, the pre-2008.1 behavior of analytic aquifer inflow equations is used. The 2008.1 behavior correctly calculates the flow from the reservoir to the aquifer in the case of connate water/gas. This change affects all analytical aquifers.
155. If set to a non-zero value, the pre-2008.1 behavior for tilted aquifer block equilibration is restored. The 2008.1 equilibration behavior uses the aquifer block properties instead of the tilted block geometrical properties for numerical aquifers associated with grid blocks of zero thickness.
156. If set to a non-zero value, the pre-2008.1 behavior for tilted block equilibration is restored. The 2008.1 equilibration behavior corrects the weighted sum calculation for tilted grid blocks, defined using corner point geometry, by adjusting the averaged inclination factors to prevent the planes representing the top and bottom faces based on these averaged inclination factors from intersecting within the grid block. This correction will typically apply to distorted grid blocks, for example, those for which one or more corners share the same or similar depth values for both the upper and lower faces.
157. If set to 1, an alternative implementation for the water density calculation algorithm is activated.
158. If set to 1, the Appleyard Chop is applied to components when they appear or disappear. This behavior is the default for simulations using the chemical reactions feature but can be disabled by setting to -1.
159. This option alters the behavior of gas accountancy, reverting various elements to pre-2008.2 behavior.
   • If set to a non-zero value divisible by 2, then if a well is set to reinject a group vapor source or a group is set to reinject a well vapor source (for example GINJGAS item 2 is set to WV) then no gas will be reinjected.
   • If set to a non-zero value divisible by 3, then make-up gas is not switched off when a group has a reinjection target set.
   • If set to a non-zero value divisible by 5, then the pre-2008.2 calculation of advance gas is used. This changes the calculation of advance gas if the WEFAC and/or GEFAC keywords are used.
   • If set to a non-zero value divisible by 7, then the pre-2008.2 calculation of availability limits is used. This changes the calculation of availability limit if the WEFAC and/or GEFAC keywords are used in conjunction with NAVAILIM.
• If set to a non-zero value divisible by 11, then the pre-2008.2 calculation of gas accounting is used. This changes the calculation of excess gas and reinjected gas if the GEFA C keyword is used.

160. If set to 1, the pre-2008.2 thermal non-linear damping is restored.

161. If set to 1 an alternative procedure for simulations with rock-filled cells is activated. Mathematically this is a null change but variations due to numerical rounding may occur.

162. If set to 1, this restores the pre-2008.1 behavior for equilibration options 2 and 3 when the reference depth is not set to GOC.

163. If set to 1, an alternative memory allocation algorithm is enabled. It reduces the number of allocations/deallocations while increasing the maximum amount of memory used.

164. If set > 0, this switch turns on Ezrokhi’s formula for the water viscosity calculation in CO2SOL/H2SSOL.

165. If set to 1, this switch turns on Ezrokhi’s formula for the water density calculation in CO2SOL/H2SSOL.

166. If set to 1, well constraints may be weakly violated in order to improve the convergence of the well model.

167. If set > 0, this switch activates the single-phase splitting initialization algorithm for GFE minimization in the flash.

168. If set to 1, the pre-2008.2 calculation of fluid in place is used, correctly accounting for satellite group injection and production fluids.

169. The default behavior is for the emulsion viscosity model to be activated in keyword WSEGSCID and deactivated in keyword WSEGSAICD. If this switch is set equal to 1, the emulsion viscosity model is deactivated in keyword WSEGSCID. If set equal to 2, the emulsion viscosity model is activated in keyword WSEGSCID. If set equal to 3, the emulsion viscosity model is both deactivated in keyword WSEGSCID and activated in keyword WSEGSAICD.

170. If set to 1, the pre-2009.1 behavior for the interpolation of the Carter-Tracy aquifer influence function tables is restored. If set to 2 the pre-2010.1 behavior for the default Carter-Tracy aquifer influence function table is restored.

171. If set to 1, the pre-2009.1 MS well energy predictor is restored.

172. If set to a non-zero value, the pre-2009.1 equilibration phase-labeling calculation is performed.

173. If set to a non-zero value, the pre-2009.1 behavior of gas accounting is used. This changes how sales and/or fuel gas is calculated if a reinjection target greater than 1 is specified.

174. If set to a non-zero value, the pre-2009.1 FACTLI calculation during solution initialization is used. The new calculation updates FACTLI based only on the single-phase cells.

175. If set to a non-zero value, the pre-2006.2 defaulting of the D-factor (item 12 of the COMPDAT keyword, or item 13 of the COMPDATL and COMPDATM keywords) is restored. A bug was fixed in this defaulting at 2006.2. Prior to that version, the completion D-factor was recalculated each time one of these keywords was present with the D-factor defaulted.

176. This option must be set in the RUNSPEC section. If set to a non-zero value the pre-2009.1 method for calculating region phase production totals (ROPT, RWPT and RGPT) is used. Pre-2009.1 these quantities were reported for the beginning of the timestep rather than the end.
177. If a multiple of 2, then the closure of history matched wells with zero production rates and banned crossflow will not automatically occur.

If a multiple of 3, then the closure of injection wells with zero injection rates and banned crossflow will not automatically occur.

If a multiple of 5, then the closure of history matched wells reverts to pre-2010.1 behavior. Specifically, reservoir volume rate controlled wells will be shut irrespective of their target rate if crossflow is banned.

178. This option controls the calculation of group production potentials. These are the sum of the well production potentials for all open wells in the group. If set to:

1 the pre-2008.3 reporting of group potentials is restored. Prior to this version, stopped well potentials were sometimes incorrectly included in the reported group potential

2 the pre-2011.1 reporting of group potentials is restored. From 2008.3 until this release, shut well potentials were sometimes incorrectly subtracted from the reported group potential.

179. Between 2009.1 and 2013.1, the derivative of the pore volume with respect to pressure in the Palmer-Mansoori rock model (see the ROCKPAMA keyword) is modified for stability. This helps with some convergence problems encountered in previous releases. The default behavior is to ignore the volumetric strain effect if it dominates the pressure effect. If a value of 1 is used, the pre-2009.1 behavior for the Palmer-Mansoori rock model is restored. If a value of 2 is used then the volumetric strain effect in the derivative is ignored in all cases. From 2013.1 this option switch has no effect unless OPTIONS3 item 295 is also set.

180. If set to a non-zero value, the pre-2009.1 behavior of allowing oil injection wells is restored.

181. If set to a non-zero value, the arguments WXMF, WYMF and WZMF of component nn in keyword ACTIONW are handled in the pre-2007.1 way: they should be written WXMFnn, WYMFnn and WZMFnn. The post-2007.1 way of writing these mnemonics is WXMF_nn, WYMF_nn and WZMF_nn.

182. If set to 1, this option ensures that the pre-2009.1 calculations for FLOOIL and FLOGAS are used for black-oil models.

183. If set to 1, this option ensures that the pre-2009.1 calculations for FLOWAT and RWFR are used for multi-component water models.

184. If set to 1, this option ensures that the pre-2009.1 calculation of well model for CO2SOL / H2SSOL case is used.

185. If set to a non-zero value, this option restores the 2008.1 behavior for equilibration option 3 when the reference depth is not set to GOC. Note that the pre-2008.1 behavior can be restored by setting item 162 of OPTIONS3 to 1.

186. This affects various behavior related to the WAVAILIM keyword.

- From 2009.1 onwards, if a well is under availability control when an unlimited injection fluid source (for example stream) is specified (see WINJGAS), then the control mode is reset to the most recent one specified explicitly for that well. The pre-2009.1 behavior (which leaves the well on availability control) can be restored by setting OPTIONS3 item 186 to a non-zero value which is a multiple of 2.

- From 2009.1 onwards, if a group is under availability control (see WAVAILIM) when more than NUPCOL Newton iterations are required, the appropriate phase availability is restored for the group from the last NUPCOL calculated value. Previously the group phase availability limit
would always have been the last NUPCOL calculated availability for the water phase. The pre-2009.1 behavior can be restored by setting OPTIONS3 item 186 to a non-zero multiple of 3.

187. If set to a non-zero value, the pre-2009.1 reporting of segment phase velocities is restored. Prior to 2009.1 the phase velocities in segments where the drift flux slip model was not used were set equal to the segment mixture velocity. Post-2009.1 the phase velocity is set to zero if the phase saturation within the segment is zero.

188. If set to a non-zero value, the pre-2009.1 calculation of temperature variation with depth in the flash calculation is restored.

189. Options for a rigorous density inversion check and correction in flash. If set to:
   -1 this switch inactivates a rigorous density inversion check for post-scalar flash.
   -2 this switch activates a rigorous density inversion check during the vector flash as well as the post-scalar flash.

190. If set to a non-zero value, the pre-2009.1 setting of well enthalpies in thermal decks is activated. From 2009.1 onwards, the well enthalpies are set whenever an action is triggered.

191. If set to a non-zero value, the pre-2009.1 density gravity calculation method for MSWs is used.

192. If set to a non-zero value, the pre-2009.1 calculation of tracer concentration is used. The calculation of tracer concentration has been changed if a well or group efficiency factor has been set (WEFAC or GEFAC).

193. If set to a non-zero value, the pre-2009.2 minimum oil saturation calculation for the Stone 1 relative permeability model with either SOMGAS or SOMWAT specified is used. This omits a correction which has been introduced from 2009.2 onwards to handle situations where end-point scaling has been specified and the critical oil-in-gas (SOGCR) or critical oil-in-water (SOWCR) saturations have been set to zero.

194. If set to a positive value, pre-2009.2 LAGTRAC calculations are recovered.
   \[\text{• If divisible by 2, streamline update and retrace times are calculated as they were for 2009.1.}\]
   \[\text{• If divisible by 3, streamline tracing uses the 2009.1 cell porosity calculation.}\]
   \[\text{• If divisible by 5, streamline tracing ignores rock compressibility.}\]
   \[\text{• If divisible by 7, streamline flowrates are calculated using the 2009.1 well/group efficiency method.}\]

195. If set to a positive value, the pre-2009.2 SORMW1 initial compositional is used. That composition consists of a SORMW1 fraction of (xi*mi) where xi is the liquid fraction and mi is the molar density.

196. If set to a non-zero value, this restores the pre-2009.2 behavior for VELDEP models using radial geometry.

197. If set to 1, the pre-2009.2 behavior for the modification of non-neighbor connections in cells which have been coarsened is restored. For the 2009.2 version onwards, the handling of NNCs across x- and y-faults is much improved, and the transmissibility of all NNCs within coarsenings may be slightly altered.

If set to 2, the pre-2011.1 behavior for the transmissibility calculations for coarse cells in which some fine cells are active and others inactive is restored. For the 2011.1 version onwards, a single inactive fine cell in a coarse cell on the boundary of a coarsening will not zero the transmissibility into the adjacent fine cell.
This option should only be used to confirm that a particular change in behavior is due to the improved modeling in 2009.2 and later.

198. For developer use.

199. If set to 1, this enables an alternative Ezrokhi calculation.

200. If set to a non-zero value, this restores the pre-2010.1 FIP (fluid-in-place) calculations for any ECLIPSE 300 data set that does not contain COAL or SOLID.

201. This item affects the method used to calculate the summary keywords BOIP, BGIP, BOIPL, BOIPG, BGIPL, BGIPG and the vectors output in the restart file when using SFIP in RPTRST. By default summary keywords and vectors are calculated using the separator defined by FIELDSEP. If set to:

1. the summary keywords and the vectors are calculated using the separators defined by FIPSEP and the regions defined by FIPNUM

2. this reverts to the pre-2010.1 behavior where the summary keywords are calculated using the separators defined by FIPSEP and the vectors are calculated using the separator defined by FIELDSEP.

202. If set to a positive value, pre-2010.1 LAGTRAC calculations are recovered.

- If divisible by 2 the pre-2010.1 method for solving LUMPING tracers is used.
- If divisible by 3 the pre-2010.1 handling of well connections is used.
- If divisible by 5 the pre-2010.1 streamline number calculation is used.

203. If set to a positive value then LUMPING tracers solved using LAGTRAC are renormalized using an alternative method. LUMPING tracers will be constrained to lie between the previous timestep solution and the un-renormalized solution for each cell, and therefore may not sum to the lumped carrier component density.

204. If set to a non-zero value, this enables the pre-2009.2 calculation method for segments whose pressure drop is defined using VFP tables.

205. If set to a positive value, this enables the pre-2009.2 behavior of ACTIONX with WLISTDYN.

206. Require that two successive Newton iterations converge.

207. If set to 1, this enables an alternative Jacobian construction for CO2SOL / H2SSOL / GASSOL.

208. If set to a non-zero value, the pre-2009.2 behavior is restored whereby the by SPECHS keyword need not be specified for thermal models with components exhibiting solid phases.

209. If set to 1, the pre-2010.1 behavior is restored. Items 4 and 5 of the PINCH keyword are disabled. The old behavior uses the new default values for these items.

210. If set to a non-zero value, the pre-2009.2 behavior of the well inflow performance relationship calculations is restored. This mainly affects Open ECLIPSE queries ending IPRA, IPRB or IPRC for wells with injecting connections.

211. If set to a non-zero value, the pre-2009.2 behavior of gas lift for wells drilled using the QDRILL keyword is used. If lift gas is specified for a closed well using WCONPROD prior to 2009.2 then no lift gas will be used if the well is opened using QDRILL.

212. If set to a positive value, the pre-2009.2 default behavior for rock volume calculation is activated. This applies to COAL Bed Methane cases that use the instant sorption model.
213. As explained in item 12 of WELSPECS, the hydrostatic heads calculated using segmented densities in the wellbore are damped to reduce the likelihood of stability problems. This is done by combining values from the previous timestep with those of the current timestep in a weighted average. An error has been identified in the implementation of this damping which meant that more damping was being applied than intended. This error has been fixed in the 2010.1 version of the simulator but has not been made the default behavior as it may result in changes in simulation results. The correct damping can be applied by setting this item to a non-zero value.

214. If set to a non-zero value, this enables the pre-2009.2 John Appleyard Linear Solver (JALS) for thermal simulations.

215. If set to 1, this enables the alternative handling of the ROCK keyword and its functionality.

216. If set to a non-zero value this enables the pre-2010.1 GRUPRIG/WDRILTIM behavior.

217. If set to a non-zero value, the pre-2010.1 behavior for thermal heavy-oil equilibration will be restored.

218. If set to a non-zero value, the pre-2010.1 thermal temperature limits will be restored.

219. If set to a non-zero value, this enables the pre-2010.1 thermal rock-filled convergence method.

220. Reserved

221. If set to a non-zero value, this enables the pre-2010.1 behavior for BDENSITY.

222. If set to a non-zero value, this enables pre-2010.1 WLISTARG target value assignments.

223. Not used at present.

224. Prior to 2010.1, the termination of fluid import to or export from a segment using the WSEGEXSS keyword (by setting the import/export type to NONE) resulted in the termination of import to or export from all segments associated with the well defined using the WSEGEXSS keyword, if the termination was applied to the first segment previously defined for import or export. From 2010.1 onwards, if the termination is applied to the first segment, the import to or export from any other segments defined using this keyword is unaffected. Pre 2010.1 behavior can be restored by setting this item to a non-zero value.

225. If set to 1, this enables an alternative saturation pressure calculation in GASWAT cases.

226. Prior to 2010.1, the output of the input oil-water capillary pressure tabular data using the EPSDEBUG or EPSDBGS keyword reported the data in GRAF format with the sign reversed as compared to the input data. This sign reversal has now been removed but may be reinstated by setting this item to a non-zero value.

227. Prior to 2010.1 well fluid composition was set to zero internally for wells with no production. From 2010.1 well fluid composition is set from the well potential solution in this situation to facilitate the later calculation of reservoir fluid volume rate targets. The pre-2010.1 behavior may be recovered by setting this item to a non-zero value.

228. If set to 1 or 3, this restores the pre-2010.1 EoS cubic root selection method for the saturation pressure calculation in flash. If set to 2 or 3, this restores the pre-2010.1 EoS cubic root selection method for the single-phase molar density calculation in flash.

229. If set to a non-zero value, the pre-2010.1 calculation of availability limits is used. This changes the calculation of availability limit if the WEFAC and/or GEFAC keywords are used in conjunction with WAVAILIM.
230. If set to a non-zero value, the pre-2010.1 use of the GEFAC keyword for transparent groups is applied. This changes group rate allocations of higher level targets if an intermediate, unconstrained group has a non-zero group efficiency factor.

231. If set to a non-zero value, the pre-2010.1 asphaltene deposition will be used. The new behavior improves the handling of the deposition model.

232. If set to 1, the pre-2010.1 density model for pure water is activated for CO2STORE. This model should not be used at higher temperatures (above 150 °C).

233. If set to a non-zero value, this enables the pre-2010.1 behavior for handling zero or negative DZ.

234. If set to a non-zero value, this enables the pre-2010.1 behavior for handling zero or negative DZ.

235. If set to a non-zero value, the pre-2010.1 default for the WELSEGS top segment effective wellbore volume is used.

236. The transmissibility and diffusivity for dual and multi porosity connections now use the simulation grid cell bulk volume and not the sum of the cell pore volume and cell rock volume. For most cases these will be equal, but for thermal and Coal Bed Methane runs, the assignment of rock volume might result in differences. The pre-2010.1 behavior can be restored by setting this item to 1.

237. The behavior when failing to converge in a thermal simulation has been improved. This includes not chopping on a minimum timestep until the maximum number of Newton iterations have been solved. The pre-2010.1 behavior can be restored by setting this item to 1.

238. If set to a non-zero value, region, production and injection rates do not include well and group efficiency factors (WEFAC and/or GEFAC) in their calculation.

239. If set to a non-zero value, the pre-2010.1 method for inferring injection gas composition from GINJGAS when using WCONINJH will be restored. In some cases, it has been found that the injection gas composition is not correctly inferred when using GINJGAS with WCONINJH and without group control. This has been rectified in 2010.1.

240. If set to a non-zero value, the pre-2010.2 version of the well solution predictor will be restored. For the 2010.2 version onwards, a correction has been made to the predictor to improve the handling of the situation when all flow terms are zero.

241. Two changes were made to the saturation pressure calculation for 2010.2: a modification to the ab-initio calculation and a change in the pressure range used for the Gibbs Free Energy minimization in the calculation. If this item is set to 1, this restores the pre-2010.2 behavior for both changes in the flash calculation. If set to 2, this only restores the pre-2010.2 ab-initio saturation pressure calculation. If set to 3, this only restores the pre-2010.2 pressure bracket for Gibbs Free Energy minimization in the saturation pressure calculation.

242. If set to 1, this restores the pre-2011.1 behavior for MULTFLT in the EDIT section. Prior to 2011.1 the fault multipliers were not properly applied to cross fault NNCs when MULTFLT was used in the EDIT section.

243. If set to 1, the pre-2010.2 version of the relative permeability hysteresis calculations, when used in combination with end-point scaling, will be restored. For the 2010.2 version onwards a correction has been made to the look-up of the maximum saturation and relative permeability table values which are used in the scaling of the hysteresis terms. This item should be specified in the RUNSPEC section.

244. If set to 1, the pre-2010.2 interpretation of the defaulting mechanism for the SWOF and SGOF / SLGOF (Family I) saturation function keywords will be restored. In some circumstances this can lead to the
incorrect construction of the **SWFN, SGFN** and **SOF3** (Family II) saturation function keywords, which are used internally for the saturation function calculations, when only one of the **SWOF** and **SGOF** / **SLGOF** keywords is defaulted for a given saturation table number. This item should be specified in the **RUNSPEC** section.

245. If set to 1, the pre-2010.2 usage of the process-dependent relative permeability and capillary pressure end-point tables **ENKRVT, ENKRVC, ENPCVT** and **ENPCVC** will be restored. This is not recommended but has been provided for backwards compatibility. For 2010.2 onwards, the internal sign of the water-oil capillary pressure has been corrected, when looked up from a process-dependent end-point table, and checks have been added to ensure that the end-point scaling arrays used to report the process-dependent end-points are allocated appropriately. This item should be specified in the **RUNSPEC** section.

246. If set to 1, the pre-2011.1 method for calculating Stone’s formula number 1 for oil relative permeability is used. This mainly affects the way end-point scaling is used with the minimum oil saturation tables (**SOMGAS** or **SOMWAT**).

247. If set to 1, the pre-2011.1 handling of **SUMMARY** keywords **AAQR** and **AAQT** is restored. Before 2011.1, the list associated with these keywords could contain an element referring to a gas analytic aquifer. If the list was empty, all analytic aquifers including gas one were listed. From the 2011.1 version on, using these keywords with a gas analytic aquifer is not allowed. Additionally, the **ALL SUMMARY** keyword implies the keywords **AAQRG** and **AAQTG**.

248. If set to 1, duplicated records in the **FAULTS** keyword are ignored. If a fault is specified twice, any multipliers associated with it are applied twice.

249. If set to 1, the pre-2011.1 interpretation of the removal of heat transfer coefficients to completions for the **WSEGHEAT** keyword will be restored. From 2011.1 onwards, the behavior has been modified to ensure that the heat transfer coefficient to the completion is removed when instructed by this keyword.

250. If set to 1, the pre-2011.1 handling of non-neighbor connections between two regions which have an associated **MULTREGT** multiplier is restored. For certain geometries, these non-neighbor connections could have the multiplier applied to them twice. Use of this option is not recommended but it is provided for backward compatibility.

251. If set to 1, the pre-2011.1 handling of **MULTPV** within an **EQUALS** keyword is restored. Use of this option is not recommended but is provided for backward compatibility. From 2011.1 onwards, the use of **MULTPV** within an **EQUALS** keyword generates an error.

252. If set to 1, the pre-2011.1 model for the calculation of water relative permeability hysteresis for water as a non-wetting phase will be restored. This change only applies when the hysteresis control keywords **EHYSTR** or **EHYSTRR** have item 2 set to a value of 7. For the 2011.1 version a correction has been applied to the calculation for the case when the current water saturation is less than the maximum historical water saturation.

253. If set to 1, the pre 2010.2 handling of inconsistent PVT table selection is restored such that, if there are any cells in the same equilibration region that use different PVT tables, this will result in a warning message only. For the 2010.2 version onwards, the default response is to issue an error message.

254. If set to 1, this activates the pre-2011.1 treatment of inline NNC transmissibilities.

255. If set to 1, the pre-2011.1 behavior is restored for the modeling of the two-phase relative permeability of oil to gas when using the **EHYSTR** or **EHYSTRR** keywords with item 2 set to 4 (Killough hysteresis applied to both wetting and non-wetting phases). In some cases it has been found that the two-phase relative permeability of oil to gas can exceed the maximum value specified in the drainage and imbibition oil saturation function tables, or the maximum value specified via the end-point scaling...
keywords KRO and IKRO (and their directional and irreversible variants) if ENDSCALE has been specified. Depending upon the choice of three-phase relative permeability model, this can result in the three-phase oil relative permeability also exceeding the maximum value from the tables or end-points. This has been addressed for the 2011.1 version by constraining the maximum value of the two-phase relative permeability of oil to gas to the maximum of the values specified in the drainage or imbibition tables or end-points. The two-phase oil relative permeability of oil to gas is combined with the two-phase oil relative permeability of oil to water to form the three-phase oil relative permeability. The constraint described here will apply to all three-phase oil relative permeability models available in ECLIPSE 300 with the exception of the IKU3P and ODD3P models.

256. If set to 1, the output generated by the RPTRST flow arguments FLOCh, FLOE, FLOOIL, FLOWAT, FLOGAS and FLORES is written to the restart file in the pre-2012.1 format, which outputs non-neighbor flows, including flows between local and global grids, in a single list. See "Restart Files" in the ECLIPSE File Formats Reference Manual for the default format. This option switch also restores the previous pre-2012.1 NNC order in the INIT file.

257. If set to 1, the pre-2011.1 method for determining whether to make an aquifer connection for a particular cell is used. This treats adjacent cells which contain a local grid as if they were inactive. If this option is not specified or set to 0, they are treated as if they were active. The new method is compatible with ECLIPSE 100.

258. If set to 1, a modified non-linear solver is used. The new non-linear solver is designed to tackle some convergence problems, such as flip-flopping. It does more work than the current default non-linear solver, and therefore will be a little slower in cases that already run well. However it can be much faster in some cases that do not run well with the default solver.

259. If set to 1, the pre-2011.1 behavior of honoring THP limits for history match wells will be used.

260. If set to 1, when using the thermal option, the surface gas density will be determined by a surface flash, rather than using the data from the DENSITY keyword.

261. If set to 1, the pre-2011.1 behavior is restored for the deselection of hysteresis via the NOHYST keyword. Prior to 2011.1, hysteresis was not deselected by this keyword if specified after the SATOPTS keyword or if WAG hysteresis was specified via the WAGHYSTR keyword.

262. If set to 1, the pre-2011.1 behavior is restored for the treatment of the PPCWMAX keyword. Prior to 2011.1, the maximum capillary pressure was not modified if this exceeded the value specified by item 1. This has now been rectified.

263. If set to 1, an alternative calculation of inter-region and inter-block phase flow rates, based on a flash at surface pressure and temperature, is activated in non-thermal CO2STORE runs.

264. If set to 1, the pre-2011.1 behavior is restored for the treatment of the WELSEGS keyword. Prior to 2011.1, in some circumstances, the redefinition of the segments in a multisegment well did not result in the re-initialization of the well. This has now been rectified.

265. If set to 1, the pre-2011.1 behavior is restored for saturation pressure calculations. In 2011.1, a new method is used to determine whether two compositions are the same. The new method is consistent with that used in the flash stability check.

266. Reserved.

267. If set to -1, the pre-2012.1 method for solving network models is used.

268. Reserved.
269. If set to 1, the pre-2012 handling of summary keywords starting with B, F, G, W with a defaulted component number is used. The current method is to store all components in the summary file. The former method stores only the last one.

270. If set to 1, the pre-2011.2 calculation of observed reservoir volume is used. The 2011.1 method did not account for NGL correctly.

271. If set to 1, the pre-2012.1 treatment for wells under WAG control is restored. For the 2012.1 version, when a well which is under WAG control is shut, the WAG control is deselected.

272. If set to 1, the pre-2012.1 treatment is restored for the defaulting behavior for the end-point scaling relative permeability keyword KRORG within a local grid when it is defaulted from the global grid.

273. If set to 1, the pre-2012.1 dual/multi-porosity diffusivity calculation is restored. (Not scaling the diffusivity with the porosity.)

274. With effect from the 2014.1 release, this item is no longer available.

275. If set to 1, the pre-2012.1 treatment is restored for the allocation of Equation of State regions for the generalized pseudo-pressure calculations (GPP). Prior to 2012.1, for models with two or more Equation of State regions, the Equation of State region number used for the calculation of the well properties associated with the GPP option was defaulted to 1.

276. If set to 1, the pre-2012.1 behavior for STEAM–GAS injection is restored to use a fixed gas molar density. In addition, the setting of this switch will also revert the improved WINJTEMP enthalpy calculation to pre-2012.1 behavior. If set to 2, the pre-2012.2 behavior for STEAM–GAS, STEAM–OIL, HCGAS and HCOIL injection is used when the surface flash detects a single phase with the wrong phase being injected. For additional information, refer to "Calculation of molar rates in ECLIPSE 300".

277. If set to 1, the pre 2012.1 treatment is restored for the output of the end-point scaling arrays to the INIT file via FILLEPS. Prior to 2012.1, the values for global grid blocks containing local grids were output as default values for end-point arrays not specified in the input data. For 2012.1 onwards, these values are now defaulted from the appropriate saturation functions.

278. If set to 1, the pre-2012.1 coal bed methane diffusion equation (for multi-porosity) is used. The coal volume will always be used.

279. If set to 1, the pre-2012.1 behavior for the reading and writing of restart records associated with the WAG hysteresis option (WAGHYSTR keyword) will be restored. For the 2012.1 version, the restart records have been restructured to permit the use of this option with parallel restarts. Additional checking of the pre-2012.1 WAG hysteresis restart records has also been introduced for 2012.1. In cases where the pre-2012.1 restart record is unsuitable, item 279 can be set to 2 to bypass the input restart record.

280. If set to 1, the pre-2012.1 behavior of wells with zero flow rate targets is used. This may return production wells with a negative production rate in some circumstances.

281. In 2012.1, the VFP table extrapolation for negative flows was changed in the network model, to provide continuous pressure drop around zero flow. If set to 1, the pre-2012.1 VFP table extrapolation in the network model is used.

282. In some cases it has been found that reading restarts which contain WAG hysteresis records created by the 2009.1 and 2009.2 versions fails with a message referring to an incorrect header record. This is associated with an attempt to read a UDA/UDQ restart record when none was defined. Under these circumstances item 282 of the OPTIONS3 keyword may be set to 1 to bypass the reading of the non-existent UDA/UDQ record thereby resulting in the correct reading of the WAG hysteresis record.
(This does not address the problems associated with the incorrect size of the WAG hysteresis which can occur in some cases with restarts written by parallel runs using versions prior to 2012.1.)

283. If set to 1, the pre-2012.1 behavior for Killough hysteresis in water-oil black oil models is restored. For the 2012.1 version, modifications have been made to take account of the critical saturation for the non-wetting imbibition saturation function when this is greater than zero and also to the modeling of the non-wetting phase hysteresis.

284. Reserved.

285. If set to 1, the pre-2012.1 behavior for modeling velocity-dependent relative permeabilities with VELDEP item 5 for the generalized pseudo-pressure option is restored. For the 2012.1 version, checks have been added to handle negative capillary numbers.

286. If set to 1, the pre-2013.1 behavior is restored for the calculation of three-phase relative permeabilities when one of the VDRP models specified by item 5 of the VELDEF keyword is selected.

For the 2013.1 version, the method of calculation of the three-phase gas and oil relative permeabilities has been modified both for the grid block properties calculation and also for VDRP effects upon the well generalized pseudo-pressure (GPP) calculation where this option has been selected. For the grid block calculation, the two-phase oil relative permeability to gas is now calculated from the two-phase gas relative permeability to oil.

For versions prior to 2013.1, the two-phase oil relative permeability to gas was calculated from the three-phase gas relative permeability. For the GPP calculation, the three-phase gas and oil relative permeabilities are now calculated in an identical manner to that employed for the grid block; this involves firstly calculating the associated two-phase relative permeability pairs and combining these to form the three-phase relative permeabilities. For versions prior to 2013.1, the three-phase oil relative permeability was calculated directly from the three-phase gas relative permeability.

287. If set to 1, the pre-2013.1 behavior is restored when modeling directional relative permeabilities without end-point scaling. Prior to 2013.1, there were some circumstances where calculations related to directional relative permeabilities assumed that end-point scaling was also being modeled.

288. Reserved.

289. Reserved.

290. If set to 1, the pre-2012.2 behavior is restored when applying default values to the KRGR, KRWR, KORG or KROWG end-points associated with the ENKRVT or ENKRVC keywords for temperature or composition dependence respectively. If set to 1, this will also restore the pre-2012.2 behavior for the reporting of the KRGR, KRWR, KORG or KROWG end-point scaling arrays in the INIT file when ENDScale and FILEEPS have been specified.

This should be specified in the RUNSPEC section.

291. If set to 1, the pre-2013.1 behavior of printing all warning messages regarding well connections to inactive grid blocks will be active. If not set, only the first five warnings will be printed and any further warnings will be suppressed.

292. If set to 1, the pre-2013.1 behavior of the algorithm solving group problems is recovered. From 2013.1 this algorithm was altered to improve robustness and speed.

293. If set to 1, the pre-2013.1 behavior of calculating drift velocities for downward sloping segments will be activated.

294. If set to 1, this reverts to the pre-2013.1 method of determining the directions of non-neighbor connections while multiplying their transmissibilities by ROCKTAB items 3 – 5, when keyword RKTRMDIR is also specified.
295. If set to 1, the pre-2013.1 ROCKPAMA / ROCKPAME calculations are used. The derivative modifications as specified by OPTIONS3 item 179 are ignored from 2013.1.

296. If set to 1, the additional WAG hysteresis regions specified via the WH2NUM and WH3NUM keywords and the associated additional processing (which applies whether these have been specified or defaulted) will be ignored. This option is only relevant when the WAGHYSTR keyword has been specified. This should be specified in the RUNSPEC section.

297. If set to 1, the pre-2013.1 version of the debug output for the scaled saturation tables specified via either the EPSDEBUG and EPSDBGS keywords will be selected.

298. Reserved.

299. If set to 1, the pre-2013.1 handling of models with both generalized pseudo-pressure and rock compaction options active will be selected.

300. If set to 1, the pre-2013.1 handling of the calculation of water molar density and water fluid-in-place will be selected, for models where SOLUBILS is included. Otherwise, the default calculation now takes into account the salt concentration in each cell and uses the corresponding brine surface density which is interpolated from SOLUBILS. Note that activating this back-compatibility option within the SCHEDULE section, and hence using a combination of both forms of water density calculation, may lead to convergence problems. It is advisable to select one calculation method for the entire simulation.

301. If set to 1, the pre-2013.1 behavior is restored for models with two-point end-point scaling where one or more of the keywords KRWR, KRGR, KROP and KROG or their imbibition counterparts have been specified. In addition, this will also revert corrections to the maximum attainable unscaled water and gas saturations.

302. If set to 1, the pre-2013.1 behavior is restored for the calculation of the oil and gas phase Forchheimer models specified via items 3 and 4 respectively of the VELDEP keyword.

303. If set to 1, the pre-2013.1 calculation of volumetric rates and fluid in place in thermal runs is selected, affecting various quantities, such as OIPG, GOR, WCT and others. It is recommended that this option is requested before the SOLUTION section in the dataset.

304. Reserved.

305. If set to 1, the pre-2013.1 calculation of equilibrium deviation terms for reactions set with EQLDREAC is restored.

306. If set to 1, the pre-2014.1 behavior will be restored for models with end-point scaling specified, where the saturations at which the end-points KRWR, KRGR, KROP and KROG (relative permeabilities at residual saturation of the displacing phase) occur are the same as the saturations at which KRW, KRG, KRO and KRO (relative permeabilities at maximum saturation) respectively occur. For the 2014.1 version, corrections have been made to prevent the maximum scaled relative permeabilities exceeding the maximum specified relative permeabilities under these circumstances. This also applies to the imbibition counterparts if hysteresis is being modeled.

307. If set to 1, warning messages will be issued for models which use the ODD3P coupled three-phase relative permeability and capillary pressure hysteresis option, where the oil-water or gas-oil primary capillary pressure curves cross the secondary (increasing) or tertiary (decreasing) hysteresis capillary pressure curves. This corresponds to pre-2013.2 behavior. If required this option should be set prior to the PROPS section.
308. If set to -1, unset LGR fracture coordinates will not be copied from the matrix even if DPGRID is specified. This was the default behavior prior to 2014.1.

309. If set to 1, this will revert to the pre-2013.2 behavior of the COMPKRIL keyword for defining user relative permeabilities for injector completions in locally refined grids.

310. If set to 1, this will revert to the pre-2013.2 behavior for the calculation of water relative permeability hysteresis in oil-wet systems. Prior to the 2013.2 version, water relative permeability hysteresis in oil-wet systems was only available for black oil reservoir models. For other models only the drainage curve was used.

311. If set to 1, the pre-2014.1 behavior for determining when a non-neighbor connection should be deleted based on its transmissibility, diffusivity and thermal transmissibility is restored. Additionally, values which are below the cutoff value for all of transmissibility, diffusivity and thermal transmissibility will be preserved provided that the NNC itself is preserved. Note that, if this option is used, the diffusivity and thermal transmissibility settings from the MINNNCT keyword will have no effect.

312. If set to 1, this will revert to the pre-2014.1 behavior for the calculation of the inverse lookup with endpoint scaling. If defaulted, inverse lookup will be performed with 3 points endpoint scaling.

313. If set to 1, this will revert to the pre-2014.1 behavior for the treatment of pressure by enumeration in cells where no value has been set, where such cells were assumed to have atmospheric pressure. If defaulted, it is an error to have an active global cell without an explicitly set pressure at the end of the SOLUTION section.

314. If set to 1, this will revert to the pre-2013.2 behavior for the modeling of water relative permeability hysteresis scanning curves for models using the WAGHYSTR keyword when end-point scaling has been specified and where the three-phase water relative permeability saturation table numbers have been specified via the WH3NUM keyword. For the 2013.1 version, in some cases, these curves were defaulted saturation tables specified via the IMBNUM keyword. This has been rectified for the 2013.2 version onwards.

315. If set to 1, this will revert to the pre-2014.1 behavior for the modeling of generalized pseudo pressure in models with more than one equation of state region. For the 2014.1 version, the generalized pseudo pressure calculations for a given well completion utilize the equation of state properties associated with the grid block containing the completion rather than those associated with the well. Prior to 2014.1, the equation of state region number associated with the well rather than the completion was used.

316. If set to 1, this will revert to the pre-2014.1 behavior for the modeling of Killough hysteresis in oil wet systems.

317. If set to 1, this will revert to the pre-2014.1 behavior for the calculation of well productivity indices. When this option is active, the effective radius derived from the connection transmissibility factor, COMPDAT item 8, may be greater than the well's drainage radius, WELSPECS item 7 in the productivity index calculation. For the ECLIPSE 100 equivalent please refer to item 226 of the OPTIONS keyword.

318. If set to 1, the preferred phase of an injection well will be updated to match the actual injection phase. This is the default behavior in ECLIPSE 100. For the ECLIPSE 100 equivalent please refer to item 227 of the OPTIONS keyword.

319. If set to 1, this will revert to the pre-2014.1 behavior for the application of the D-Factors for wells or well completions for which values have been specified via the WDFAC keyword, the WDFACCOR keyword or item 12 of the COMPDAT keyword (or item 13 of the COMPDATL keyword). For the 2014.1 version, an additional check has been added to avoid negative discriminants in the solution of
the quadratic in the flow rate used to calculate the D-Factor multiplier. In all other respects, the
calculation itself has not been changed.

320. If set to 1, this will revert to the pre-2014.1 behavior for the defaulting of the hysteresis control
parameters when hysteresis has been specified via the HYSTER option of the SATOPTS keyword in
the RUNSPEC section but the EHYSTR keyword has been omitted from the PROPS section. For
versions prior to 2014.1, neither relative permeability or capillary pressure hysteresis were selected
correctly if, in addition to the EHYSTR keyword, none of the alternative keywords used for specifying
hysteresis, for example, HYKR, HYP, HYS, HYSTJ or HYSTK were specified. For the 2014.1
version, if hysteresis has been specified via the HYSTER option of the SATOPTS keyword but the
EHYST keyword has been omitted, the hysteresis parameters will be assigned to the default values
specified for the EHYSTR keyword and any alternative definitions associated with alternative
hysteresis keywords will be superseded.

321. If set to 1, this will revert to the pre-2014.1 behavior for the specification of negative capillary
pressure end-points via the PCW and PCG keywords and their imbibition counterparts. Prior to 2014.1,
if negative values were specified, the values actually used would default to the corresponding
saturation table end-point values. For the 2014.1 version, if negative PCW or PCG values are specified,
these will be used instead of the saturation table end-point values. This change of behavior also applies
to capillary pressure end-points calculated via the JFUNC and JFUNCR keywords.

322. If set to 1, the non-linear residual checks will not be output by the JALS linear solver.

Examples

The examples show two ways to set items 21 and 34. The first example sets them on one line by defaulting
the first 20 items, setting 21, defaulting the next 12 items and setting 34:

```
  * OPTIONS3
  20* 2  12* 1 /
```

The second example sets the switches on two lines, by defaulting 20 items and setting item 21, then
defaulting 33 items and setting item 34:

```
  OPTIONS3
  20* 2 /
  OPTIONS3
  33* 1 /
```
**OPTLOAD**

**Load optimization from a given iteration**

This keyword is used to hot-start a reservoir optimization simulation using the optimizer control parameter values from a specified optimization iteration.

In order to use this keyword, you must have a optimization state file, which contains the information from the reservoir optimizer at the required iteration. This file must be in the same directory as the simulation model file that uses the OPTLOAD keyword.

**CAUTION:** There is no guarantee of cross-version compatibility of the optimization state file. It is possible that an optimization state file generated by one version of the simulator will not load into another version of the simulator.

The optimization state file is output by the optimizer during each optimization iteration.

- The rootname of the optimizer state file is the rootname of the original optimization simulation file.
- The file extension for the optimization state file depends on the output requirements specified from the original optimization. The file extensions are shown in the table below:

<table>
<thead>
<tr>
<th>Extension</th>
<th>Unified</th>
<th>Formatted</th>
</tr>
</thead>
<tbody>
<tr>
<td>FOPT</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>OPT</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>K000#</td>
<td>No</td>
<td>Yes</td>
</tr>
<tr>
<td>J000#</td>
<td>No</td>
<td>No</td>
</tr>
</tbody>
</table>

*Table 3.24: Optimization state file extensions*

**Note:** When the optimizer is hot-started, only the control parameter values from the original optimization iteration are used, not the gradient values or line-search parameters. This is because any changes to the optimization problem (such as modifying the objective function, constraints or control parameters) between the original optimization and the hot-started optimization invalidates the stored optimization gradients.

**Example**

This example shows how to restart an optimization using a previous optimization run from optimization iteration number 8. This example assumes that the required optimization state file has the rootname BASECASE and is in the same directory as the new (hot-starting) simulation model.

```
OPTLOAD 'BASECASE' 8/
```
Specify ODD3P relative permeability and capillary pressure hysteresis model options

This keyword is used to specify options that control the operation of the ODD3P three-phase relative permeability and capillary pressure coupled hysteresis model.

The keyword is followed by a single record consisting of two integer items. The record is terminated with a slash (/):

1. Option for selecting which ODD3P model to use for the calculation of three-phase relative permeabilities and capillary pressures when the ODD3P option has been specified.
   1. 2008.1 version of ECLIPSE 300 ODD3P option.
   2. 2008.2 version of ECLIPSE 300 ODD3P option.
   3. 2009.1 version of ECLIPSE 300 ODD3P option.
   4. Current version of ECLIPSE 300 ODD3P option.

   DEFAULT: 4.

   For the 2008.1 and earlier versions it has been found that, in some cases, the switching behavior when switching from primary to hysteresis curves was incorrect, in that the wrong hysteresis curve was selected. This has been rectified for the 2008.2 version onwards, which is used by default. However, the 2008.1 version of ODD3P has been retained for backwards compatibility.

   For the 2009.1 version, some amendments have been made to the calculation of the derivatives associated with the relative permeability calculations. The 2008.2 version, which does not include these amendments, may be specified by setting this item to 2.

   For the current version, some amendments have been made to the calculation of the derivatives associated with the scaling of the end-point saturations. The 2009.1 version, which does not include these amendments, may be specified by setting this item to 3.

2. Option for selecting which set of saturations is used to determine changes in saturation direction.
   1. Grid block saturations.
   2. Normalized saturations.

   DEFAULT: 2.

   For very small changes in saturations, instantaneous differences in saturation direction between the grid block saturations and the ODD3P normalized saturations for a given phase can occur. Given that the ODD3P model utilizes the normalized saturations for the hysteresis calculations, consistency is assured by using the normalized saturations for the detection of saturation direction changes and this method is used by default. The detection of saturation direction changes based upon grid block saturations is available for backwards compatibility. This was the only method available for the 2008.1 version of ODD3P.

   This keyword is only relevant if the ODD3P three-phase relative permeability and capillary pressure hysteresis model has been selected and will otherwise be ignored. If this keyword is not specified with the ODD3P option, the default options described above will be selected automatically.
Example

The example below selects the current ODD3P model and saturation direction switching based upon normalized saturations both of which are the defaults.

OPTODD3P
  2  2 /

Keywords

1583
OPTOPTS

Specify optimization options

This keyword is used to specify options that control the operation of the optimizer.

The keyword is followed by a single record consisting of three integers. The record is terminated with a slash (/):

1. Flag indicating which method is to be used to calculate the search direction in the optimizer. A value of 1 indicates that the steepest descent method is to be used, while a value of 2 indicates that the conjugate gradient method is to be used.

   DEFAULT: 1

2. Flag indicating which method is to be used to determine the free/dependent parameter split used when handling active constraints.

   A value of 1 indicates that the first parameters in the list should be taken as the dependent parameters at each time period - this was the behavior pre-2006.1. The use of this method means that the performance of the optimizer can be dependent on the order that the parameters are specified in the OPTPARS keyword.

   A value of 2 indicates that the dependent parameters should be chosen as the ones for which the constraints have the greatest sensitivity at each time period. The use of this method should mean that the optimization is not dependent on the order that the parameters are specified in the OPTPARS keyword.

   DEFAULT: 2

3. Flag indicating which formulation is to be used for calculating the Lagrangian gradients.

   In the original formulation, as described in Introducing Additional Constraints in the ECLIPSE Technical Description is used to eliminate $\Psi$. In the alternative formulation, as described in Introducing Additional Constraints in the ECLIPSE Technical Description is used to eliminate $\Phi$.

   A value of 1 indicates that the (more expensive) alternative formulation should only be used when necessary, and the original formulation should be used otherwise. item 11 of the OPTTUNE keyword is used to decide when the switch between the two formulations is necessary.

   A value of 2 indicates that the original formulation should always be used.

   A value of 3 indicates that the alternative formulation should always be used.

   A value of 4 indicates that the method used in the previous versions of the code should be used. This is the same as original formulation but with an attempt made to continue the optimization if insensitive constraints are encountered. This formulation is only present for back-compatibility. The mechanism used to continue the optimization has no mathematical or physical foundation and may only work for a small number of cases. It has been entirely superseded by the alternative formulation.

   DEFAULT: 1

Example

This example shows how to change the optimizer to use the conjugate gradient method, and also to use the specified OPTPARS parameter order for choosing the dependent parameters when handling constraints.

```
OPTOPTS
  2  1  /
```
**OPTPARS**

**Specify parameters for optimization**

This keyword is used to specify the control parameters to be varied in an optimization run of the simulator.

The keyword must be followed by a set of records, each specifying a parameter to be used in the optimization. Each record should be terminated with a slash (/) and the set of records must end with a slash. At least one record is required to perform an optimization.

This keyword should only appear once in the data file.

1. **Control parameter.**

   Available control parameters are:
   - **FOPR** Field oil production rate
   - **FWPR** Field water production rate
   - **FWIR** Field water injection rate
   - **FGPR** Field gas production rate
   - **FGIR** Field gas injection rate
   - **FLPR** Field liquid production rate
   - **GOPR** Group oil production rate
   - **GWPR** Group water production rate
   - **GWIR** Group water injection rate
   - **GGPR** Group gas production rate
   - **GGIR** Group gas injection rate
   - **GLPR** Group liquid production rate
   - **WBHP** Well bottom hole pressure
   - **WOPR** Well oil production rate
   - **WWPR** Well water production rate
   - **WWIR** Well water injection rate
   - **WGPR** Well gas production rate
   - **WGIR** Well gas injection rate
   - **WLPR** Well liquid production rate

   **SCSA** Segment cross sectional area (for parameterization of multisegment well segments defined using the `WSEGVALV` keyword). The segment cross sectional area parameter is currently only available for producing multisegment wells, not injectors.

---

**Keywords**

1585
2. Unique identifier for the domain to which the parameter specified in the first item applies. This should be the name of a well or group that has already been defined in the SCHEDULE section. It is not possible to use wildcard characters to specify more than one well or group. Well lists, specified using the WLIST keyword, cannot be used as the domain identifier.

   If the parameter is specific to a segment of a multisegment well (such as a specific valve control) then the segment number should also be provided in the form NAME:XX where NAME denotes the well name and XX the segment number. See the examples for more information.

3. The minimum permissible value of the control parameter during the optimization.
   DEFAULT: 0.0

4. The maximum permissible value of the control parameter during the optimization
   DEFAULT: 1.0E+20

5. Index of first report time at which this parameter may be varied.
   DEFAULT: 0 (or the first report index in a RESTART case)

6. Index of last report time at which this parameter may be varied.
   DEFAULT: The last report index requested in the SCHEDULE section.

7. Control mode to be used between the start and end report times of this parameter. This is only used for SCSA parameters (see item 1).
   Since a well can only have one control mode, then if multiple SCSA parameters are specified on the same well at the same time, then the first control mode specified will be used for the well.

   Available control modes are:

   ORIG  Original control from the base case
   OIL   Oil rate control
   WAT   Water rate control
   GAS   Gas rate control
   LIQ   Liquid rate control
   BHP   Bottom hole pressure control
   THP   Tubing head pressure control. (This control mode is only available if the well has an associated VFP table.)

   DEFAULT: ORIG

Notes

- The initial value of a parameter is taken as the value of the corresponding simulated value in the optimization base case simulation. For example, if a \texttt{WBHP} parameter is specified on a particular well, then the initial value for that parameter at each report period in the optimization is taken as the actual bottom hole pressure for that well in the base simulation.
The SCSA parameter is a special case here, because there is no concept of an “SCSA control mode” for a well. When an SCSA parameter is used, the well control mode can be specified in item 7. The target for the specified SCSA parameter control mode during optimization runs is taken as the actual value from the optimization base simulation, and all other targets and limits are removed from the well.

- If the ‘ORIG’ SCSA control mode is specified, then the well control (for example from the WCONPROD keywords) from the base case is kept for the optimization runs.
- The specified SCSA parameter well control mode will be ignored if there is a well parameter on the same well at the same time. This is because the well parameter specifies the control mode and this takes precedence over the segment SCSA control mode.
- When a parameter is specified, all other limits that might interfere with this are removed by the optimizer so that it has complete control over the effect of the parameter on the simulation. For example, if a WOPR parameter is specified for a particular well, then during the optimization simulations, all other limits (on rates, pressures and so forth) are removed and the well is removed from responding to any higher level group control. If other limits are required during optimization, then these should be specified using the OPTCONS keyword.

Examples

Example 1

```bash
--- Bottom hole pressure control parameters
OPTPARS
'WBHP' P1 /  
'WBHP' INJE1 /  
'WBHP' INJE2 10.0 100.0 2 7/  
'WBHP' INJE3 2* 10 20/  
/
```

Example 2

```bash
--- Multisegment valve control parameters
--- MSPROD:12 refers to segment 12 of the well named MSPROD
OPTPARS
'SCSA' MSPROD:29 0.0001 0.05/  
'SCSA' MSPROD:12 0.0001 0.05/  
/
```
**OPTTUNE**

### Specify optimization tuning parameters

This keyword is used to specify tuning parameters for an optimization run of the simulator. The keyword is optional.

The keyword is followed by a single record consisting of some or all of the following items.

1. **The initial constraint tolerance used in the optimization run.**
   - DEFAULT: 0.05
2. **The minimum constraint tolerance.**
   - DEFAULT: 0.05
3. **The number of optimization iterations to allow for progression from the initial to the minimum constraint tolerance.**
   - DEFAULT: 10
4. **Objective function convergence tolerance between optimization iterations.**
   - DEFAULT: 1.0e-06
5. **Minimum objective function RMS gradient.**
   - DEFAULT: 1.0e-06
6. **Parameter convergence tolerance between optimization iterations.**
   - DEFAULT: 1.0e-06
7. **Parameter boundary relative tolerance.**
   - DEFAULT: 1.0e-03
8. **Line search alpha convergence tolerance.**
   - DEFAULT: 1.0e-06
9. **Line search alpha minimum.**
   - DEFAULT: 1.0e-03
10. **Line search alpha maximum.**
    - DEFAULT: 1.0
    - Note: Increase this number for a more aggressive optimization line search strategy.
11. **The tolerance used to decide if the active constraints are insensitive to the dependent parameters. (This is only used if item 3 of **OPTOPTS** is set to 1.)**
    - DEFAULT: 1.0e-06

### Example

```
OPTTUNE 0.10 1* 20 /
```

---

### Keywords

- ECLIPSE
  - 100
  - 300
- SPECIAL
  - RUNSPEC
  - GRID
  - EDIT
  - PROPS
  - REGIONS
  - SOLUTION
  - SUMMARY
  - SCHEDULE
- OPTIMIZE
ORTHERROR  Set a minimum orthogonality error for calculation of nine-point transmissibilities

This keyword is used to declare a threshold orthogonality error to employ nine-point transmissibilities whenever the MPFA keyword is used to select the nine-point MPFA scheme.

See "Tensor permeability" in the ECLIPSE Technical Description for further information.

The keyword should be followed by a single positive real number, the dimensionless minimum orthogonality error, in order to calculate nine-point transmissibilities for an interface.

DEFAULT: 0.000000

The data must be terminated by a slash (/).

The keyword causes any interfaces with an orthogonality error less than the input value to use the normal five-point transmissibilities.

Example

```
ORTHERROR
  0.001 /
```
ORTHERRX  Minimum orthogonality error for X-interface nine-point transmissibility calculation

This keyword is used to declare a threshold orthogonality error for interfaces in the X-direction to employ nine-point transmissibilities whenever the MPFA keyword is used to select the nine-point MPFA scheme.

See "Tensor permeability" in the ECLIPSE Technical Description for further information.

The keyword should be followed by a single positive real number, the dimensionless minimum orthogonality error in order to calculate nine-point transmissibilities for an X-direction interface.

DEFAULT: 0.000000

The data must be terminated by a slash (/).

The keyword causes any interfaces in the X-direction cell with an orthogonality error less than the input value to use the normal five-point transmissibilities.

Example

```
ORTHERRX
  0.001 /
```
**ORTHERRY**  Minimum orthogonality error for Y-interface nine-point transmissibility calculation

This keyword is used to declare a threshold orthogonality error for interfaces in the Y-direction to employ nine-point transmissibilities whenever the MPFA keyword is used to select the nine-point MPFA scheme.

See "Tensor permeability" in the *ECLIPSE Technical Description* for further information.

The keyword should be followed by a single positive real number, the dimensionless minimum orthogonality error, in order to calculate nine-point transmissibilities for a Y-direction interface.

**DEFAULT:** 0.000000

The data must be terminated by a slash (/).

The keyword causes any interfaces in the Y-direction cell with an orthogonality error less than the input value to use the normal five-point transmissibilities.

**Example**

```
ORTHERRY
0.001 /
```
ORTHERRZ  Minimum orthogonality error for Y-interface nine-point transmissibility calculation

This keyword is used to declare a threshold orthogonality error for interfaces in the Z-direction to employ nine-point transmissibilities whenever the MPFA keyword is used to select the nine-point MPFA scheme.

See "Tensor permeability" in the ECLIPSE Technical Description for further information.

The keyword should be followed by a single positive real number, the dimensionless minimum orthogonality error in order to calculate nine-point transmissibilities for a Z-direction interface.

DEFAULT: 0.000000

The data must be terminated by a slash (/).

The keyword causes any interfaces in the Z-direction cell with an orthogonality error less than the input value to use the normal five-point transmissibilities.

Example

| ORTHERRZ | 0.001 / |

Keywords

1592
OUTRAD

Outer radius for radial geometry

The keyword should be followed by a single positive real number denoting the outer radius of the reservoir, terminated by a slash (/).

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

If the outer radius is specified using OUTRAD, the program calculates a set of cell radii (or R-direction thicknesses), which observe the correct inner radius (see the INRAD keyword), outer radius and number of cells in the R-direction.

In addition, the keyword DRV may be used to specify the R-direction thicknesses of some of the cells. In this case, the program honors these input values and just calculates the DRV values of the remaining cells.

If an incomplete set of DRV values has been entered, the missing radii are supplied in a geometric series when OUTRAD is specified.

For instance, if there are NR cells in the R-direction, and i_L is the index of the first block for which the DRV value has not explicitly been entered, then the outer radii for blocks 1 to i_L-1 are obtained by successively accumulating the sum of the DRV s and INRAD.

For the remaining blocks (that is, for i > i_L-1), the outer radii of two neighboring blocks in the radial direction (the i^{th} and i-1^{th} grid blocks) will be related by

\[ \frac{R_i}{R_{i-1}} = \left( \frac{\text{OUTRAD}}{R_{i_l}} \right)^{1/(NR-(i_L-1))} \]

so that neighboring pairs of radii have the same ratio for all i > i_L-1. DR for the i^{th} cell is then given by

\[ DR_i = R_i - R_{i-1} \]

If only INRAD and OUTRAD are specified, and no DRV values have been specified, then the grid block radii are calculated from the above formulae with i_L = 1 and R_0 defined equal to INRAD.

The following restrictions apply to the use of OUTRAD:

- It should only be used with radial geometry.
- OUTRAD should not be specified when the grid is defined by corner-point geometry using the keyword COORD.
- OUTRAD must be greater than INRAD.
- It is ignored if a complete set of DRV values (NR entries) has been input.

Example

With NR=5:

```
INRAD
  0.2 /
DRV
  1.0 3.0 /
OUTRAD
  500 /
```

Keywords

1593
OUTSOL

Controls solution output for GRAF and flexible restarts

See keyword RPTRST, which has superseded the OUTSOL keyword.

Note: Although the old OUTSOL keyword is superseded, it is still recognized for back compatibility. If a data set contains both OUTSOL and RPTRST, the later one will determine the output to the Restart file.
**Initial overburden boundary conditions**

This keyword specifies initial overburden boundary conditions when initial stress calculation INIT is specified with the keyword `GEOMECH` in the RUNSPEC section.

This keyword allows initial overburden on the top face. An overburden at the top face is a force/unit area. If this keyword is not specified, the overburden is defaulted to zero, which means the top face is free to move unless otherwise specified.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A vertical overburden on Z+ face.
   - **UNITS**: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
   - **DEFAULT**: 0

2. First block to be modified in the X direction
   - **DEFAULT**: 1 if an X- face

3. Last block to be modified in the X direction
   - **DEFAULT**: NX if an X+ face

4. First block to be modified in the Y direction
   - **DEFAULT**: 1 if a Y- face

5. Last block to be modified in the Y direction
   - **DEFAULT**: NY if a Y+ face

**Example**

```plaintext
---Set a 3000 psi overburden
OVERBDI
=3000 /
```
OVERBDIL

Initial overburden boundary conditions in an LGR

This keyword specifies initial overburden boundary conditions for an LGR when initial stress calculation INIT is specified with the keyword GEOMECH in the RUNSPEC section.

This keyword allows initial overburden on top face in a local grid refinement. An overburden at top face is a force/unit area. If this keyword is not specified, the overburden is defaulted to be zero which means the top face is free to move, unless otherwise specified.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Name of the Local Grid Refinement
2. A vertical overburden on Z+ face.
   
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
   
   DEFAULT: 0
3. First block to be modified in the X direction
   
   DEFAULT: 1 if an X- face
4. Last block to be modified in the X direction
   
   DEFAULT: NX if an X+ face
5. First block to be modified in the Y direction
   
   DEFAULT: 1 if a Y- face
6. Last block to be modified in the Y direction
   
   DEFAULT: NY if a Y+ face

Example

```plaintext
-- For the local grid refinement LGR1 set a 3000 psi overburden
OVERBDIL
  'LGR1' -3000 /
/
```

Keywords

1596
OVERBURD  Rock overburden pressure tables

The data comprises NTROCC (see item 2 in keyword ROCKCOMP in the RUNSPEC section) tables of rock overburden data, each terminated by a slash (/). Each table consists of 2 columns of data:

1. The depth.
   Values should increase monotonically down the column.

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

2. The overburden pressure values should increase monotonically down the column.

   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see item 4 of keyword TABDIMS in the RUNSPEC section).

The NTROCC records correspond to the rock compaction table region numbers, which are set using keyword ROCKNUM.

The OVERBURD keyword can be used if the rock compaction option is selected (keyword ROCKCOMP in RUNSPEC). The overburden pressure at the grid block depth is subtracted from the pore pressure to look up the pore volume and transmissibility multiplier values. If the keyword is not present then the overburden pressure is taken to be zero.

ECLIPSE by default tabulates the rock compaction multipliers against fluid pressure. But if the OVERBURD keyword is specified, the ROCKTAB, ROCKTABH or ROCK2D and ROCK2DTR keywords tabulate the compaction multipliers against the effective fluid pressure instead (P\text{eff fl} = \text{P}_\text{fluid} - \text{Overburden}). If the overburden pressure is greater than the fluid pressure, as it usually is, then the effective fluid pressure (or stress) is negative. This leads to non-intuitive compaction tables with negative effective pressures in the first column.

The compaction tables can be input in a more intuitive manner by selecting the ‘STRESS’ option in the ROCKOPTS keyword before the ROCKTAB, ROCKTABH or ROCK2D and ROCK2DTR keywords. In this case the pore volume and transmissibility multipliers are tabulated against the effective overburden pressure,

\[ P\text{eff ov} = \text{Overburden} - \text{P}_\text{fluid}, \text{ which is } - P\text{eff fl}. \]

See also keywords ROCKTAB, ROCKTABH, ROCK2D, ROCK2DTR and ROCKOPTS, and “Rock compaction” in the ECLIPSE Technical Description.

**Example**

With NTROCC=1 and NPPVT ≥5:

```
OVERBURD
  2000  600
  4000 1200
  6000 1800
  8000 2300
 10000 2800
```

**Keywords**
**OVERTR**

**Overwrite radial-direction transmissibilities**

This keyword specifies a value to be overwritten on to the transmissibilities during the SCHEDULE section, for flow in the radial direction.

The value is applied to the transmissibility between the cell and its neighbor in the appropriate positive direction.

The default is to supply a value for each cell in the grid, that is each keyword should be followed by \( N_R \cdot N_\theta \cdot N_z \) values in natural order, with the R index changing fastest.

If values are defaulted or set negative, the transmissibilities for the corresponding cells are not modified.

These keywords will normally be used with **BOX** and **ENDBOX**, when the number of values should be the number of cells in the current **BOX**.

The form of the input is analogous to that for the **OVERTX** keyword.

**Example**

Set transmissibility values in a box to zero:

```
-- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  5  5  3  8  4  6  /
OVERTR
  18*0.0  /
ENDBOX
```

The modified transmissibilities can be output to the Print file by using the mnemonic **MULT** in the **RPTSCHED** keyword.

**Note:** It is important to note that if the transmissibilities are also modified using transmissibility multiplier **MULTR**, the resulting output takes into account both of these actions.
**OVERTTHT**

**Overwrite azimuthal-direction transmissibilities**

This keyword specifies a value to be overwritten on to the transmissibilities during the SCHEDULE section, for flow in the azimuthal direction.

The value is applied to the transmissibility between the cell and its neighbor in the appropriate positive direction.

The default is to supply a value for each cell in the grid, that is each keyword should be followed by $N_R \cdot N_\theta \cdot N_z$ values in natural order, with the R index changing fastest.

If values are defaulted or set negative, the transmissibilities for the corresponding cells will not be modified.

These keywords are normally used with BOX and ENDBOX, when the number of values should be the number of cells in the current BOX.

The form of the input is analogous to that for the **OVERTY** keyword.

**Example**

Set transmissibility values in a box to zero:

```
-- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1  4  3  4  6 /
OVERTTHT
  12*0.0 /
ENDBOX
```

The modified transmissibilities can be output to the Print file by using the mnemonic MULT in the RPTSCHED keyword.

**Note:** It is important to note that if the transmissibilities are also modified using transmissibility multiplier MULTTHT, the resulting output takes into account both of these actions.
Overwrite X-direction transmissibilities

This keyword specifies a value to be overwritten on to the transmissibilities during the SCHEDULE section, for flow in the X direction.

The value is applied to the transmissibility between the cell and its neighbor in the appropriate positive direction.

UNITS: cP·rm³/day/bars (METRIC), cP·rb/day/psi (FIELD), cP·rcc/hr/atm (LAB), cP·rm³/day/atm (PVT-M)

The default is to supply a value for each cell in the grid, that is each keyword should be followed by \( N_x \cdot N_y \cdot N_z \) values in natural order, with the X index changing fastest.

If values are defaulted or set negative, the transmissibilities for the corresponding cells will not be modified.

These keywords is normally used with BOX and ENDBOX, when the number of values should be the number of cells in the current BOX.

Example

Set transmissibility values in a box to 18.6:

```plaintext
-- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX
  23  25  13  15  1  8 /
OVERTX
  72*18.6 /
ENDBOX
```

The modified transmissibilities can be output to the Print file by using the mnemonic MULT in the RPTSCHED keyword.

**Note:** It is important to note that if the transmissibilities are also modified using transmissibility multiplier MULTX, the resulting output will take into account both of these actions.
**Overwrite Y-direction transmissibilities**

This keyword specifies a value to be overwritten on to the transmissibilities during the `SCHEDULE` section, for flow in the Y direction.

The value is applied to the transmissibility between the cell and its neighbor in the appropriate positive direction.

**UNITS:** \( \text{cP}\cdot\text{m}^3/\text{day}/\text{bars (METRIC)}, \text{cP}\cdot\text{rb}/\text{day}/\text{psi (FIELD)}, \text{cP}\cdot\text{rcc}/\text{hr}/\text{atm (LAB)}, \text{cP}\cdot\text{rm}^3/\text{day}/\text{atm (PVT-M)} \)

The default is to supply a value for each cell in the grid, that is each keyword should be followed by \( N_x \cdot N_y \cdot N_z \) values in natural order, with the X index changing fastest.

If values are defaulted or set negative, the transmissibilities for the corresponding cells will not be modified.

These keywords is normally used with `BOX` and `ENDBOX`, when the number of values should be the number of cells in the current `BOX`.

**Example**

Set transmissibility values in a box to 1.4:

```
-- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX
  23  25  13  15  1  8 /
OVERTY
  72*1.4 /
ENDBOX
```

The modified transmissibilities can be output to the Print file by using the mnemonic `MULT` in the `RPTSCHED` keyword.

**Note:** It is important to note that if the transmissibilities are also modified using transmissibility multiplier `MULTY`, the resulting output takes into account both of these actions.
Overwrite Z-direction transmissibilities

This keyword specifies a value to be overwritten on to the transmissibilities during the SCHEDULE section, for flow in the Z direction.

The value is applied to the transmissibility between the cell and its neighbor in the appropriate positive direction.

UNITs: cP.m³/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm³/day/atm (PVT-M)

The default is to supply a value for each cell in the grid, that is each keyword should be followed by $N_x \cdot N_y \cdot N_z$ values in natural order, with the X index changing fastest.

If values are defaulted or set negative, the transmissibilities for the corresponding cells will not be modified.

These keywords are normally used with BOX and ENDBOX, when the number of values should be the number of cells in the current BOX.

Example

Set transmissibility values in a box to 2.0:

```plaintext
--- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX  23 25 13 15 1 8 /
OVERTZ 72*2.0 /
ENDBOX
```

The modified transmissibilities can be output to the Print file by using the mnemonic MULT in the RPTSCHED keyword.

Note: It is important to note that if the transmissibilities are also modified using transmissibility multiplier MULTZ, the resulting output takes into account both of these actions.
This section contains an alphabetic listing of keywords starting with the letter P.
Component parachors

In a run with $N_c$ components, this keyword associates a parachor with each component. These need only be entered if surface tensions are to be calculated (MISCIBLE option requested in RUNSPEC).

The keyword should be followed by $N_c$ values.

The use of parachors is discussed in "Evaluation of surface tensions" in the ECLIPSE Technical Description.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

UNITS: (dynes/cm)$^{1/4}$cc/gm-M (METRIC), (dynes/cm)$^{1/4}$cc/gm-M (FIELD), (dynes/cm)$^{1/4}$cc/gm-M (LAB), (dynes/cm)$^{1/4}$cc/gm-M (PVT-M)

DEFAULT: There are no component-wise default values. The users are recommended not to default these to the internal default value.

Example

```
PARACHOR
74.92 192.74 390.4 /
```
**Select a parallel run**

Initiates a parallel run and sets options for domain decomposition.

This keyword should be followed by up to two items of data, followed by a slash (/).

1. The number of domains (NDMAIN).
   
   **DEFAULT:** 1

2. The type of run.
   
   **SERIAL** For the serial implementation of the parallel algorithm.
   
   This activates the ECLIPSE 300 WARP solver
   
   **DISTRIBUTED** For distributed memory machines.
   
   **DEFAULT:** DISTRIBUTED

If the second item in the PARALLEL keyword is set to SERIAL, the parallel algorithm is executed on a single processor, without any message passing. Consequently no parallel speedup is observed. This option is for testing only, not for production runs.

See "Parallel option" in the *ECLIPSE Technical Description*.

**Example**

```
PARALLEL
   --NDMAIN MACHINE TYPE
     4 DISTRIBUTED /
```
PARAOPTS  Options affecting domain decomposition

The keyword is used to control a number of options that may improve performance in parallel runs.

The keyword should be followed by one record. The record contains one character string and a number of integers that allow some tuning of the parallel domain decomposition option (see “Options to improve performance” in the ECLIPSE Technical Description). The data should be terminated with a slash (/).

1. Method used for global reduction operations.
   Either TREE or T caused a hypercube algorithm to be used for global sums and maximum operations.
   Any other character string will cause the data to be collected on one processor and the sum computed there.
   DEFAULT: TREE

2. Determines whether parallel code generates one Print (PRT) file for the run or one per slave. The same applies to debug (DBG), log (LOG) and message (MSG) files. The log file contains output that would otherwise be output to the screen on the master process. If one file is produced for each type some output from processes other than process 1 may be lost.
   0 produces one file of each stated type. However, if one or several slave processes encounter an error or a bug condition, or a stop limit imposed by keyword MESSAGES, the files for each process will be created. Note that these files may not contain some initial information, in which case setting this item to 1 as explained below is still useful.
   1 produces a file of each stated type for each process. No log file is produced for process 1 as this is the output that is sent directly to the screen.
   DEFAULT: 0

3. Sets the size (in Megabytes) for the message passing buffers used in the MPI version.
   ECLIPSE 100 will always use at least about 9.5 Megabytes. Therefore, values less than 10 will have no effect.
   DEFAULT: 0

4. Sets the number of buffers to be used for sending messages in the MPI version.
   ECLIPSE 100 will always use at least 2 buffers.
   DEFAULT: 2

5. Values greater than 0 may reduce the memory required for tracer calculations. Currently disabled.
   DEFAULT: 0

6. Values greater than 0 cause coarsenings that cross domain boundaries to be split, and the cells in the shared plane left uncoarsened.
   DEFAULT: 0

7. Values greater than 0 cause the inlining of NNCs in the outer solver direction to be turned off.
   DEFAULT: 0

8. Values greater than 0 cause the reports in the PRT file controlled by the RPT keywords to be produced. Not producing these reports saves memory and decreases execution time.

Keywords
DEFAULT: 0

9. Reserved.

Example

```
PARAOPTS
  2* 10 3 /
```
PARTTRAC  Dimensions for the partitioned tracer option

The data consists of three items, giving data needed for the partitioned tracer option. This keyword must be used to activate the option. The option is available if either the environmental tracer option or the surfactant option is used (see "Environmental tracers" and "Surfactant model" in the ECLIPSE Technical Description).

The data must be terminated by a slash (/).

1. **NPARTT**: The maximum number of partitioned tracers, entered using keyword TRACER in the PROPS section.
   - DEFAULT: 0
2. **NKPTMX**: The number of K(P) tables describing the partitioning, entered using keywords TRACERKP or TRACERKM in the PROPS section.
   - DEFAULT: 0
3. **NPKPMX**: The maximum number of pressure points in each K(P) table, entered in keyword TRACERKP.
   - DEFAULT: 0

**Example**

```
PARTTRAC
  1  5  30 /
```

**Note**

Partitioned tracers allow ECLIPSE 100 to solve a range of problems that could not be tackled otherwise. For example, if marked gas is injected into a reservoir it is possible that the marker may dissolve in the water; the partitioned tracer option can model this behavior. With a single phase tracer in, say, the gas phase, ECLIPSE solves a conservation equation for the total amount of tracer in a grid block, taking into account the inflow and outflow of gas. With a partitioned tracer, one or more additional phases can be nominated (in the TRACER keyword) into which the tracer may also dissolve. Please see the "Environmental tracers" chapter in the ECLIPSE Technical Description.

**Note**: This keyword cannot be used with two-phase tracers.
PATHS

Pathname aliases

This keyword allows aliases for pathname roots to be defined, to facilitate transfer between different working directories and to avoid unnecessarily long pathnames. The aliases can be used with the INCLUDE, IMPORT, RESTART and GDFILE keywords. See "File handling in ECLIPSE" in the ECLIPSE Technical Description.

The keyword should be followed by up to 100 records, each of which is terminated by slash (/). The list of aliases is terminated by a null record (that is a record with no data before the terminating slash). Each record consists of the following items of data.

1. ALIAS: Alias for the pathname or pathname root
   (up to 8 characters long and enclosed in quotes).
2. PATHNAME: Full pathname or pathname root to be substituted for the alias
   (up to 132 characters long and enclosed in quotes).

When using an alias in INCLUDE, IMPORT, RESTART or GDFILE, it should be prefixed by a ‘$’ which identifies the following string as an alias. ECLIPSE then substitutes the string $ALIAS by PATHNAME.

Example

```--Runspec section
PATHS
  'GRIDS' '/disk1/studies/grids' /
  'HMATCH' '/disk1/pred/match' /
  'VFP' '/disk2/wells/vfp' /
/
---Grid section
INCLUDE
  '$GRIDS/case1.grdecl' /
--Props section
IMPORT
  '$HMATCH/endpoints/model1.dat' /
/```
**Initial bubble point pressure**

This keyword should only be used in a black oil run. The keyword should be followed by one real number for every grid block, specifying the initial bubble point pressure. The data field should be terminated by a slash (/).

**UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to \( NDIVIX \)), followed by the Y axis index (from 1 to \( NDIVIY \)), and finally the Z axis index (from 1 to \( NDIVIZ \)). \( NDIVIX, NDIVIY \) and \( NDIVIZ \) are specified in RUNSPEC keyword **DIMENS**. Repeat counts may be used for repeated values (for example 115*2.893). Note that spaces must not be inserted on either side of the asterisk (*).

This keyword is an alternative to the **RS** keyword for defining the initial dissolved gas distribution in runs with enumerated initial conditions.

When using explicit initialization, you must ensure that the initial solution is stable and physically reasonable.

See also the keywords **PRESSURE**, **PRVD**, **RV**, **PDEW**, **SWAT** and **SGAS** in the SOLUTION section.

**Example**

With \( NDIVIX=8, NDIVIY=6 \) and \( NDIVIZ=2 \) specified in RUNSPEC keyword **DIMENS**:

```
PBUB
  48*3042
  48*3063 /
```

---

**Keywords**

1610
Bubble point versus depth tables for equilibration

The data comprises NTEQUL (see item 1 in keyword EQLDIMS) tables of bubble point pressure versus depth, one for each equilibration region. Each table consists of 2 columns of data, and must be terminated by a slash (/). Column:

1. Depth values.
   The values should increase monotonically down the column.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

2. The corresponding values of the bubble point pressure (P_{bub}).
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NDRXVD (see item 3 in keyword EQLDIMS).

This keyword is an alternative to the RSVD keyword, in which the dissolved gas-oil ratio is tabulated against depth for each equilibration region. See also keyword EQUIL.

The PBVD keyword cannot be used in API tracking runs.

See "Initializing the study" in the ECLIPSE Technical Description for further information.

Example

With NTEQUL=1 and NDRXVD ≥3:

<table>
<thead>
<tr>
<th>Depth</th>
<th>Pressure</th>
</tr>
</thead>
<tbody>
<tr>
<td>7000</td>
<td>4000</td>
</tr>
<tr>
<td>8000</td>
<td>4020</td>
</tr>
</tbody>
</table>
| 9000    | 4045     | /
**PCG, IPCG**

**Scaled maximum gas capillary pressures**

This keyword allows the maximum gas-oil capillary pressure to be scaled on a grid block by grid block basis. The keyword can only be used when the end-point scaling option is active (keyword `ENDSCALE` in the RUNSPEC section).

The capillary pressure is given by:

\[ P_c = P_{ct} \left( \frac{PCG}{P_{cm}} \right) \]

Eq. 3.138

where

- \( P_{ct} \) Capillary pressure from the table
- \( P_{cm} \) Maximum \( P_c \) in the table - at \( S_g = 1 - S_{wco} \)
- PCG Maximum \( P_c \) from the PCG data.

The PCG keyword should be followed by one real number for each grid block in the current input box, specifying the maximum gas capillary pressure value. The data must be terminated by a slash (/).

**UNITS:** bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the PCG keyword is omitted, the maximum gas \( P_c \) value defaults to the value used in the appropriate gas saturation function table.

If the hysteresis model is active (switch `HYSTER` in keyword `SATOPTS`), then the PCG data will scale the drainage curve only. The imbibition curve can be scaled by using the equivalent IPCG keyword.

If the ODD3P relative permeability and capillary pressure hysteresis model is selected, this keyword will be ignored.

The PCG keyword should not be used in combination with the J-function option (keyword `JFUNC`). In this case, the J-function calculation always defines the water capillary pressure scaling and the input PCG array is ignored.

See also keywords `SGFN` and `SGOF` in the PROPS section.

Refer to "Saturation table scaling" in the ECLIPSE Technical Description for more information.

**Note:** The PCG data used to scale the capillary pressure must be consistent with the capillary pressure from the table. In particular PCG has to have the same sign as \( P_{cm} \) in order to avoid having a change in the wettability. If PCG and \( P_{cm} \) have different signs (for example PCG negative and \( P_{cm} \) positive) the scaling will not be performed. Moreover no scaling will be performed if \( P_{cm} \) is zero (see item 74 in OPTIONS for a different scaling).
Example

With ENDSCALE specified in RUNSPEC; and NDIVIX=1, NDIVIY=7 and NDIVIZ=3 specified in RUNSPEC keyword **DIMENS**):

\[
\begin{align*}
\text{PCG} \\
6*10.0 \\
6*12.0 \\
6*15.0 & /
\end{align*}
\]
PCG32D

2D table of gas/oil capillary pressure

This keyword together with keyword SGF32D is an alternative to keyword SGFN. It allows the three-phase gas/oil capillary pressure to be input directly as a function of oil and water saturations in a two-dimensional table. The gas relative permeability must be provided using keyword SGF32D.

The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) tables of gas/oil capillary pressure, tabulated against both oil and water saturations, each terminated by a slash (/).

Each table consists of a number of records:

1. The oil saturation values

Values should be between 0 and 1 and should increase monotonically. The number of oil saturation values should not exceed NSSFUN (see item 3 of keyword TABDIMS).

Subsequent records (a maximum of NSSFUN records):

• The water saturation.

• Followed by the gas/oil capillary pressure values for this water saturation at all the corresponding oil saturations entered in Record 1.

The gas/oil capillary pressures should be monotonically decreasing. The number of gas/oil capillary pressure values should not exceed the number of oil saturation values entered in Record 1.

UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

The columns of data must also comply with the following constraints:

• The water saturation (the first item in the second and subsequent records) should increase monotonically down the column. The gas/oil capillary pressure for each oil saturation should decrease monotonically down the column.

• Default values may not be placed within the table.

The entire table may be defaulted provided that the table is not the first. Defaulted tables are replaced with a copy of the previous table.

The keyword cannot be used with the end-point scaling, hysteresis or miscible options.

Example

With NTSFUN=1 and NSSFUN ≥11:

```
PCG32D
   -----------------------------------SOIL-----------------------------------
   0.01 0.27 0.32 0.37 0.42 0.47 0.52 0.57 0.72 0.77 0.78 /
   --SWAT
   0.22 4.000 3.800 3.500 3.000 2.500 2.000 1.500 1.000 0.500 0.200 0.000/
   0.25 0.700 0.600 0.500 0.400 0.300 0.200 0.100 0.050 0.030 0.000 /
   0.30 0.600 0.500 0.400 0.300 0.200 0.100 0.090 0.030 0.000 /
   0.35 0.500 0.400 0.300 0.200 0.100 0.050 0.030 0.001 0.000 /
   0.40 0.400 0.300 0.200 0.100 0.060 0.030 0.001 0.000 /
   0.50 0.300 0.200 0.100 0.002 0.000 0.000 /
   0.65 0.200 0.100 0.000 0.00 /
   0.80 0.100 0.000 /
   1.00 0.000 /
   /
```
PCODD3P  Capillary pressure calculation control for ODD3P option

The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) tables of capillary pressure calculation control data, each terminated by a slash (/). Only one row per table should be entered. This keyword is only used with the ODD3P relative permeability and capillary pressure hysteresis option and is optional.

Each table consists of a single row containing 7 columns of data:

1. Capillary pressure functional dependence calculation control.
   DEFAULT: 6
   One of thirteen functional dependence relationships may be selected for the calculation of capillary pressure. These are numbered 0 to 12, with a value of 0 used to set the capillary pressures to zero. These functional dependences are described below.

2. Constant $\delta_g$ in gas saturation dependent capillary pressure weighting function.
   DEFAULT: 0.0
   This parameter is only used if the value of the functional dependence calculation control parameter defined in column 1 is assigned a value between 1 and 8 inclusive. The use of these weighting functions is described below.

3. First exponent $\alpha_g$ in gas saturation dependent capillary pressure weighting function.
   DEFAULT: 1.0
   This parameter is only used if the value of the functional dependence calculation control parameter defined in column 1 is assigned a value between 1 and 8 inclusive. The use of these weighting functions is described below.

4. Second exponent $\beta_g$ in gas saturation dependent capillary pressure weighting function.
   DEFAULT: 1.0
   This parameter is only used if the value of the functional dependence calculation control parameter defined in column 1 is assigned a value between 1 and 8 inclusive. The use of these weighting functions is described below.

5. Constant $\delta_w$ in water saturation dependent capillary pressure weighting function.
   DEFAULT: 0.0
   This parameter is only used if the value of the functional dependence calculation control parameter defined in column 1 is assigned a value between 1 and 8 inclusive. The use of these weighting functions is described below.

6. First exponent $\alpha_w$ in water saturation dependent capillary pressure weighting function.
   DEFAULT: 1.0
   This parameter is only used if the value of the functional dependence calculation control parameter defined in column 1 is assigned a value between 1 and 8 inclusive. The use of these weighting functions is described below.
7. Second exponent $\beta_w$ in water saturation dependent capillary pressure weighting function.

DEFAULT: 1.0

This parameter is only used if the value of the functional dependence calculation control parameter defined in column 1 is assigned a value between 1 and 8 inclusive. The use of these weighting functions is described below.

The capillary pressure weighting functions $H(S_g)$ and $F(S_w)$ are calculated as shown below:

$$H(S_g) = \frac{\delta_g S_g^{\alpha_g}}{S_g^\beta_g + 1 - S_g^{\beta_g}}$$

and

$$F(S_w) = \frac{\delta_w S_w^{\alpha_w}}{S_w^\beta_w + 1 - S_w^{\beta_w}}$$

The representative capillary pressures, that is, the values obtained from table look-up using the normalized saturations are corrected according to the following equations:

$$P_{cow} = P_{cow}^\ast + HR$$
$$P_{cgo} = P_{cgo}^\ast + FR$$
$$R = P_{cgw}^\ast - P_{cgo}^\ast - P_{cow}^\ast$$

where the capillary pressure terms $P_{ci}^\ast$ denotes the representative capillary pressures obtained from table look-up as a function of normalized hysteresis saturation $\hat{S}_i$ or $\hat{S}_j$. The capillary pressures may be made functions of one of two normalized saturations as shown below:

<table>
<thead>
<tr>
<th>Saturation Dependence</th>
<th>Oil-Water System</th>
<th>Gas-Oil System</th>
<th>Gas-Water System</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$P_{cow}(\hat{S}_o)$</td>
<td>$P_{cgo}(\hat{S}_g)$</td>
<td>$P_{cgw}(\hat{S}_g)$</td>
</tr>
<tr>
<td>2</td>
<td>$P_{cow}(\hat{S}_w)$</td>
<td>$P_{cgo}(\hat{S}_o)$</td>
<td>$P_{cgw}(\hat{S}_w)$</td>
</tr>
</tbody>
</table>

Table 3.25: Saturation dependencies

The functional dependence calculation control parameter defined above in column 1 of the `PCDODD3P` keyword selects the method used to calculate the capillary pressures as shown below:

<table>
<thead>
<tr>
<th>Relationship type</th>
<th>Calculation of oil-water and gas-oil capillary pressure</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>$P_{cow} = 0.0$</td>
</tr>
<tr>
<td>1 and 2</td>
<td>$P_{cow} = P_{cow}^\ast + HR$</td>
</tr>
<tr>
<td>3 and 4</td>
<td>$P_{cow} = P_{cow}^\ast + HR$</td>
</tr>
<tr>
<td>5 and 6</td>
<td>$P_{cow} = P_{cow}^\ast + HR$</td>
</tr>
<tr>
<td>7 and 8</td>
<td>$P_{cow} = P_{cow}^\ast + HR$</td>
</tr>
</tbody>
</table>
saturations (S-Weighted Model). This model has been developed by StatoilHydro ASA see [Ref. 104].

Calculation of oil-water and gas-oil capillary pressure

<table>
<thead>
<tr>
<th>Relationship type</th>
<th>Calculation of value R</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>( R = \hat{P}(\tilde{S}<em>g) \cdot \hat{P}</em>{cgo}(\tilde{S}<em>g) \cdot \hat{P}</em>{cow}(\tilde{S}_o) )</td>
</tr>
<tr>
<td>2</td>
<td>( R = \hat{P}_{cgw}(\tilde{S}<em>w) \cdot \hat{P}</em>{cgo}(\tilde{S}<em>g) \cdot \hat{P}</em>{cow}(\tilde{S}_o) )</td>
</tr>
<tr>
<td>3</td>
<td>( R = \hat{P}_{cgw}(\tilde{S}<em>g) \cdot \hat{P}</em>{cgo}(\tilde{S}<em>o) \cdot \hat{P}</em>{cow}(\tilde{S}_o) )</td>
</tr>
<tr>
<td>4</td>
<td>( R = \hat{P}_{cgw}(\tilde{S}<em>w) \cdot \hat{P}</em>{cgo}(\tilde{S}<em>o) \cdot \hat{P}</em>{cow}(\tilde{S}_o) )</td>
</tr>
<tr>
<td>5</td>
<td>( R = \hat{P}_{cgw}(\tilde{S}_g) \cdot \hat{P}(\tilde{S}<em>g) \cdot \hat{P}</em>{cow}(\tilde{S}_o) )</td>
</tr>
<tr>
<td>6</td>
<td>( R = \hat{P}_{cgw}(\tilde{S}<em>w) \cdot \hat{P}</em>{cgo}(\tilde{S}<em>o) \cdot \hat{P}</em>{cow}(\tilde{S}_w) )</td>
</tr>
<tr>
<td>7</td>
<td>( R = \hat{P}_{cgw}(\tilde{S}<em>g) \cdot \hat{P}</em>{cgo}(\tilde{S}<em>o) \cdot \hat{P}</em>{cow}(\tilde{S}_w) )</td>
</tr>
<tr>
<td>8</td>
<td>( R = \hat{P}_{cgw}(\tilde{S}<em>w) \cdot \hat{P}</em>{cgo}(\tilde{S}<em>o) \cdot \hat{P}</em>{cow}(\tilde{S}_w) )</td>
</tr>
</tbody>
</table>

Table 3.26: Functional relationships for the calculation of capillary pressure

R is calculated according to the value of the capillary pressure functional dependence calculation control parameter (column 1) for values between 1 and 8 above as shown below:

<table>
<thead>
<tr>
<th>Relationship type</th>
<th>Calculation of value R</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>( P_{cow} = \hat{P}<em>{cow}(\tilde{S}<em>o) ) ( P</em>{cgo} = \hat{P}</em>{cgw}(\tilde{S}<em>w) \cdot \hat{P}</em>{cow}(\tilde{S}_o) )</td>
</tr>
<tr>
<td>10</td>
<td>( P_{cow} = \frac{S_g}{S_g + S_o} \hat{P}<em>{cgw}(\tilde{S}<em>w) + \frac{S_o}{S_g + S_o} \hat{P}</em>{cow}(\tilde{S}<em>w) ) ( P</em>{cgo} = \frac{S_w}{S_o + S_w} \hat{P}</em>{cgw}(\tilde{S}<em>g) + \frac{S_o}{S_o + S_w} \hat{P}</em>{cgo}(\tilde{S}_g) )</td>
</tr>
<tr>
<td>11</td>
<td>( P_{cow} = \frac{S_g}{S_g + S_o} \hat{P}<em>{cgw}(\tilde{S}<em>w) + \frac{S_o}{S_g + S_o} \hat{P}</em>{cow}(\tilde{S}<em>w) ) ( P</em>{cgo} = \hat{P}</em>{cgo}(\tilde{S}_g) )</td>
</tr>
<tr>
<td>12</td>
<td>( P_{cow} = \frac{S_o(S_o[\hat{P}<em>{cgw}(\tilde{S}<em>w) - \hat{P}</em>{cgo}(\tilde{S}<em>g)] + S_w[\hat{P}</em>{cgw}(\tilde{S}<em>w) - \hat{P}</em>{cgo}(\tilde{S}<em>g)] + S_o(S_o + S_w)\hat{P}</em>{cow}(\tilde{S}<em>w)}{S_o(S_o + S_w)} ) ( P</em>{cgo} = \left[ \frac{S_o}{(S_o + S_w)} + \frac{S_w}{S_o + S_w}(\frac{S_gS_o}{S_g + S_o + S_w}) \right] \cdot \hat{P}</em>{cgo}(\tilde{S}<em>g) + \left[ \frac{S_gS_w}{(S_g + S_o + S_w)} \right] \cdot \hat{P}</em>{cgw}(\tilde{S}<em>w) - \left[ \frac{S_w}{(S_g + S_o + S_w)} \right] \cdot \hat{P}</em>{cow}(\tilde{S}_w) )</td>
</tr>
</tbody>
</table>

Table 3.27: Functional relationship types and the calculation of R

Capillary pressure functional dependence calculation control parameter options 9, 10 and 11 correspond to cases where R is equal to 0.0. For options 10, 11 and 12 the saturation weightings are obtained from the grid block saturations \( \tilde{S}_i \). Option 12 incorporates saturation weighting based upon all three grid block saturations (S-Weighted Model). This model has been developed by StatoilHydro ASA see [Ref. 104].
If the PCODD3PG keyword is specified, the parameters in columns 2, 3 and 4 which are used to calculate the function \( H(S_g) \) are ignored and the values of this function are obtained from the tabulated data specified by the keyword PCODD3PG.

Similarly, if the PCODD3PW keyword is specified, the parameters in columns 5, 6 and 7 which are used to calculate the function \( F(S_w) \) are ignored and the values of this function are obtained from the tabulated data specified by the keyword PCODD3PW.

The values assigned to items 2 to 7 and the optional superseding of these values by the specification of the keywords PCODD3PG and PCODD3PW are only relevant if, for a given saturation region, the capillary pressure functional dependency (item 1) has been set to an option between 1 and 8.

Default values (represented by 1*) may be inserted as required in columns 1 to 7. When the table is read in, these defaults are replaced by the default values specified above.

The second and subsequent tables may be defaulted entirely. Defaulted tables are replaced with a copy of the previous table.

Only the tables associated with the primary saturation region numbers defined using the PSTNUM keyword are used. Tables associated with the secondary and tertiary hysteresis saturation region numbers defined using the ISTNUM and DSTNUM keywords respectively are ignored and may be defaulted.

See "Saturation functions" in the ECLIPSE Technical Description for a description of the method used to compute the oil relative permeability in blocks where all three phases are present.

See also the keywords EPSODD3P, PCODD3PG and PCODD3PW.

**Example**

With NTSFUN=3

```
PCODD3PG
6 1.0 0.1 0.2 1.0 0.11 0.3 /
9 6*/
2 1.3 0.1 0.2 1.5 0.13 0.34 /
```
PCODD3PG  Capillary pressure calculation gas saturation function

The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) tables of capillary pressure calculation gas saturation functions, each terminated by a slash (/). This keyword is only used with the ODD3P relative permeability and capillary pressure hysteresis option and is optional. If specified, this keyword replaces the gas saturation function \( H(S_g) \) defined by the parameters specified in the keyword PCODD3P with the tabulated function defined here.

Each table consists of 2 columns of data:

1.  The gas saturation.
   Values should be between 0.0 and 1.0 inclusive and should increase monotonically down the column. The first and final values should be 0.0 and 1.0 respectively.

2.  The corresponding values for the function \( H(S_g) \).
   Values should be between 0.0 and 1.0 and should be level or increasing down the column. The first and final values should be 0.0 and 1.0 respectively.

   Default values (represented by 1*) may be inserted as required in column 2. When the table is read in, defaults are replaced by values computed by linear interpolation.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see item 3 of keyword TABDIMS).

The entire table may be defaulted provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

Only the tables associated with the primary saturation region numbers defined using the PSTNUM keyword are used. Tables associated with the secondary and tertiary hysteresis saturation region numbers defined using the ISTNUM and DSTNUM keywords respectively are ignored and may be defaulted.

See "Saturation functions" of the ECLIPSE Technical Description for a description of the method used to compute the oil relative permeability in blocks where all three phases are present.

See also the keywords EPSODD3P, PCODD3P and PCODD3PW.

**Example**

With NTSFUN=1 and NSSFUN ≥ 8:

```
PCODD3PG
0.0000  0.000
0.2000  0.030
0.3000  0.050
0.4000  0.150
0.5000  0.240
0.8000  0.670
0.9000  0.800
1.0000  1.000  /
```
**PCODD3PW  Capillary pressure calculation water saturation function**

The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) tables of capillary pressure calculation water saturation functions, each terminated by a slash (/). This keyword is only used with the ODD3P relative permeability and capillary pressure hysteresis option and is optional. If specified, this keyword replaces the water saturation function $F(S_w)$ defined by the parameters specified in the keyword PCODD3P with the tabulated function defined here.

Each table consists of 2 columns of data:

1. The water saturation.
   
   Values should be between 0.0 and 1.0 inclusive and should increase monotonically down the column. The first and final values should be 0.0 and 1.0 respectively.

2. The corresponding values for the function $F(S_w)$.
   
   Values should be between 0.0 and 1.0 and should be level or increasing down the column. The first and final values should be 0.0 and 1.0 respectively.

   Default values (represented by 1*) may be inserted as required in column 2. When the table is read in, defaults are replaced by values computed by linear interpolation.

   There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see item 3 of keyword TABDIMS).

   The entire table may be defaulted provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

   Only the tables associated with the primary saturation region numbers defined using the PSTNUM keyword are used. Tables associated with the secondary and tertiary hysteresis saturation region numbers defined using the ISTNUM and DSTNUM keywords respectively are ignored and may be defaulted.

   See "Saturation functions" in the ECLIPSE Technical Description for a description of the method used to compute the oil relative permeability in blocks where all three phases are present.

   See also the keywords EPSODD3P, PCODD3P and PCODD3PG.

**Example**

With NTSFUN=1 and NSSFUN≥8:

```
PCODD3PW
0.0000  0.000
0.2000  0.030
0.3000  0.050
0.4000  0.150
0.5000  0.240
0.8000  0.670
0.9000  0.800
1.0000  1.000  /
```
Critical pressures

In a run with \( N_c \) components, this keyword associates a critical pressure with each component.

The keyword should be followed by \( N_c \) values.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword `TABDIMS`. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

**UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atm (PVT-M)

**DEFAULT:** There are no component-wise default values. The users are recommended not to default these to the internal default value.

**Example**

For a four-component system:

```
PCRIT
1050 1300 1500 1803 /
```
Critical pressures for detailed components

In a lumped simulation run with $N_d$ detailed components, this keyword associates a critical pressure with each detailed component. Lumped simulations are enabled with the LUMPDIS keyword in the RUNSPEC section; further details can be found in "Delumping and lumping" in the ECLIPSE Technical Description.

The keyword should be followed by a record of $N_d$ values, terminated with a slash, for each reservoir equation of state region. The values represent the critical pressure of each detailed component, in the order in which the detailed components are defined in the LUMPING keyword.

If multiple equations of state are being used, a record of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

UNITs: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

Example

For a system with four detailed components and one EoS region:

```plaintext
PCRITDET
1050 1300 1500 1803 /
```
Critical pressures for surface EoS

This optional keyword supplies critical pressure values for use in the surface calculations. The default, if this is not supplied, is for these to follow the reservoir values supplied with PCRIT.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.
Critical pressures for surface EoS for detailed components

This optional keyword supplies critical pressure values for use in the surface calculations for the detailed components in a lumped simulation. The default, if this is not supplied, is for these to follow the reservoir values supplied with PCRITDET. Lumped simulations are enabled with the LUMPDIMS keyword in the RUNSPEC section; further details can be found in "Delumping and lumping" in the ECLIPSE Technical Description.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.
**PCW, IPCW**  

**Scaled maximum water capillary pressures**

This keyword allows the maximum water-oil (or water-gas) capillary pressure to be scaled on a grid block by grid block basis. The keyword can only be used when the End Point Scaling option is active (keyword `ENDSCALE` in the RUNSPEC section).

The capillary pressure is given by:

\[ P_c = P_{ct} \left( \frac{P_{cW}}{P_{cm}} \right) \]

*Eq. 3.139*

where

- \( P_{ct} \) Capillary pressure from the table
- \( P_{cm} \) Maximum \( P_c \) in the table - at \( S_w = \) connate water
- \( P_{cW} \) Maximum \( P_c \) from the \( P_{cW} \) data.

The \( P_{cW} \) keyword should be followed by one real number for each grid block in the current input box, specifying the maximum water capillary pressure value. The data must be terminated by a slash (/).

**UNITS:** bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the \( P_{cW} \) keyword is omitted, the maximum water \( P_c \) value defaults to the value used in the appropriate water saturation function table.

If the Hysteresis model is active (switch 'HYSTER' in keyword `SATOPTS`), then the \( P_{cW} \) data scales the drainage curve only. The imbibition curve can be scaled by using the equivalent IPCW keyword.

**Note:** The \( P_{cW} \) keyword should not be used in combination with the J-function option (keyword `JFUNC`). In this case, the J-function calculation always defines the water capillary pressure scaling and the input \( P_{cW} \) array is ignored. The maximum capillary pressure for each cell is reported to the INIT file. This is shown as the PCW property in the 3D Viewer.

If the ODD3P relative permeability and capillary pressure hysteresis model is selected, this keyword will be ignored.

See also keywords `SWFN`, `SWOF` and `SWATINIT` in the PROPS section.

Refer to "Saturation table scaling" in the *ECLIPSE Technical Description* for more information.

**Note:** The \( P_{cW} \) data used to scale the capillary pressure must be consistent with the capillary pressure from the table. In particular PCW has to have the same sign as \( P_{cm} \) in order to avoid having a change in the wettability. If PCW and \( P_{cm} \) have different sign (for example PCW negative and \( P_{cm} \) positive) the scaling will not be performed. Moreover no scaling will be performed if \( P_{cm} \) is zero (see item 74 of `OPTIONS` and item 143 of `OPTIONS3` for a different scaling).
Example

With ENDSCALE specified in RUNSPEC; and NDIVIX=1, NDIVIY=7 and NDIVIZ=3 specified in RUNSPEC keyword **DIMENS**:

```
PCW
6*78.0
6*20.0
6*150.0 /
```
PCW32D  

2-D table of water/oil capillary pressure

This keyword together with keyword SWF32D is an alternative to keyword SWFN. It allows the three-phase water/oil capillary pressure to be input directly as a function of oil and gas saturations in a two dimensional table. The water relative permeability must be provided using SWF32D keyword.

The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) tables of water/oil capillary pressure, tabulated against both oil and gas saturations, each terminated by a slash (/).

Each table consists of a number of records:

1. The oil saturation values
   
   Values should be between 0 and 1 and should increase monotonically. The number of oil saturation values should not exceed NSSFUN (see item 3 of keyword TABDIMS).

Subsequent records (a maximum of NSSFUN records):

   • The gas saturation.
   
   Followed by the water/oil capillary pressure values for this gas saturation at all the corresponding oil saturations entered in Record 1.

   The water/oil capillary pressures should be monotonically increasing, from an initial value of zero. The number of water/oil capillary pressure values should not exceed the number of oil saturation values entered in Record 1.

UNITs: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

The columns of data must also comply with the following constraints:

   • The gas saturation (the first item in the second and subsequent records) should increase monotonically down the column. The water/oil capillary pressure for each oil saturation should increase monotonically down the column.

   • Default values may not be placed within the table.

The entire table may be defaulted provided that the table is not the first. Defaulted tables are replaced with a copy of the previous table.

The keyword cannot at present be used with the end-point scaling, hysteresis or miscible options.

Example

With NTsfUN=1 and NSSFUN≥11:

```plaintext
PCW32D
-----------------------------------SOIL-----------------------------------
  0.01  0.27  0.32  0.37  0.42  0.47  0.52  0.57  0.72  0.77  0.78 /
--SGAS
  0.00   0.000 1.000 2.000 3.000 4.000 5.000 6.000 7.000 8.000 9.000 10.00/ 
  0.05   0.100 1.100 2.100 3.100 4.100 5.100 6.100 7.100 8.100 9.100 / 
  0.10   0.200 1.200 2.200 3.200 4.200 5.200 6.200 7.200 8.200 / 
  0.15   0.300 1.300 2.300 3.300 4.300 5.300 6.300 7.300 8.300 / 
  0.20   0.400 1.400 2.400 3.400 4.400 5.400 6.400 7.400 / 
  0.25   0.500 1.500 2.500 3.500 4.500 5.500 6.500 / 
  0.30   0.600 1.600 2.600 3.600 4.600 5.600 / 
  0.35   0.700 1.700 2.700 3.700 4.700 / 
  0.40   0.800 1.800 2.800 3.800 / 
/ 
```
Initial dew point pressure

This keyword should only be used in a black oil run. The keyword should be followed by one real number for every grid block, specifying the initial vapor oil-gas ratio. The data field should be terminated by a slash (/).

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*0.00377). Note that spaces must not be inserted on either side of the asterisk (*).

This keyword is an alternative to the RV keyword for defining the initial vaporized oil distribution in runs with enumerated initial conditions.

When using explicit initialization, you must ensure that the initial solution is stable and physically reasonable.

See also the keywords PRESSURE, PRVD, RS, PBUB, SWAT and SGAS in the SOLUTION section.

Example

With NDIVIX=8, NDIVIY=6 and NDIVIZ=2 specified in RUNSPEC keyword DIMENS:

```
PDEW
 48*6123
 48*6098 /
```

Keywords

1628
Select a division of the x-direction for parallel runs

This is an optional keyword associated with the PARALLEL keyword. It specifies the placement of the parallel domain decomposition in the X-direction. It is recommended that the alternative more general keyword PSPLITX be used instead of PDIVX.

The keyword is followed by N
\textsubscript{px} integer items (where N
\textsubscript{px} is set using the NPROCX keyword). Each integer specifies the last x-index for the particular processor. The last item must have the value N\textsubscript{x}.

See also "Parallel option" in the ECLIPSE Technical Description.

Example

\begin{verbatim}
DIMENS 128 128 22 /
PARALLEL 10 /
NPROCX 4/
   -- processor1 x value range 1-10
   -- processor2 x value range 11-30
   -- processor3 x value range 31-44
   -- processor4 x value range 45-128
PDIVX 10 30 44 128 /
\end{verbatim}
Select a division of the Y-direction for parallel runs

This is an optional keyword associated with the `PARALLEL` keyword. It specifies the placement of the parallel domain decomposition in the Y-direction. It is recommended that the alternative, more general, keyword `PSPLITY` be used instead of `PDIVY`.

The keyword is followed by \(N_y\) integer items (where is set using the `NPROCY` keyword). Each integer specifies the last y-index for the particular processor. The last item must have the value \(N_y\).

See also "Parallel option" in the *ECLIPSE Technical Description*.

**Example**

```
DIMENS 128 128 22 /
PARALLEL 10 /
NPROCY 4/
   -- processor1 y value range 1-10
   -- processor2 y value range 11-30
   -- processor3 y value range 31-44
   -- processor4 y value range 45-128
PDIVY 10 30 44 128 /
```
Dew point versus depth tables for equilibration

The data comprises NTEQUL (see item 1 in keyword EQLDIMS) tables of dew point pressure versus depth, one for each equilibration region.

Each table consists of 2 columns of data, and must be terminated by a slash (/).

1. Depth values.
   The values should increase monotonically down the column.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
2. The corresponding values of the dew point pressure (P\textsubscript{dew}).
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NDRXVD (see item 3 in keyword EQLDIMS).

This keyword is an alternative to the RVVD keyword, in which the vaporized oil-gas ratio is tabulated against depth for each equilibration region. See also keyword EQUIL.

See "Initializing the study" in the ECLIPSE Technical Description for further information.

Example

With NTEQUL=1 and NDRXVD\geq3:

```
 PDVD
  7000  3000
  8000  3020
  9000  3045 /
```
**PEBI**

**Specifies a Pebi/Petra grid is to be used**

This keyword instructs ECLIPSE that an unstructured Pebi/Petra grid is used. The grid is expected to be generated by FloGrid.

The keyword is followed by the following items, terminated with a slash (/).

**Note:** ECLIPSE will not itself generate grid files from a simulation that includes the `PEBI` keyword, so they should not be deleted unless the user is prepared to regenerate them from the original source.

1. Flag to allow negative transmissibilities. (ECLIPSE 100 only.)
   - **YES** Allow negative transmissibilities
   - **NO** Disallow negative transmissibilities
   
   **DEFAULT:** **NO**

2. Flag to avoid grid calculations in ECLIPSE, and to perform data checks appropriate to unstructured grid data.
   - **YES** Assume the pore volumes and transmissibilities are all supplied and skip attempting to calculate them
   - **NO** Perform pore volume and transmissibility calculations
   
   **DEFAULT:** **NO**

**Note:** ECLIPSE 300 does not support any Pendi grid that includes `DUALPORO` or `DUALPERM`.

**Example**

```
PEBI
NO YES /
```
The data in this keyword consists of a row for each PENUM region in the model. Each row consists of at least eight items but may contain values for other items if they are not defaulted. These items describe the coefficients in each PENUM region for the Petro-elastic Model. The first ten items are real numbers and the last item is an integer.

Each row of data must be terminated by a slash (/). Empty rows (containing just their terminating slash) default to a copy of the previous row.

1. The salinity of water, S, in the region.
   **UNITS:** ppm (METRIC), ppm (FIELD), ppm (LAB)
   This is used in the calculation of bulk modulus of brine in the Petro-elastic Model.

2. The temperature, T, in the region.
   **UNITS:** °C (METRIC), °F (FIELD), °C (LAB)
   This is used in the calculation of the bulk modulus of oil, water and gas in the Petro-elastic Model. A negative value is not permitted for this item.

3. The combined density of the minerals (for example sand, shale, silt) in the region.
   **UNITS:** kg/m$^3$ (METRIC), lb/ft$^3$ (FIELD), gm/cc (LAB)

4. The first order coefficient of effective porosity, $\phi_{eff}$, in the calculation of the bulk modulus of the mineral material in the region. This is the $C_1$ term in equation 3.140:
   \[ K_{minerals} = C_1\phi_{eff} + C_0 \]  
   **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB)

5. The zero$^{th}$ order coefficient of effective porosity, $\phi_{eff}$, in the calculation of the bulk modulus of the mineral material in the region. This is the $C_0$ term in equation 3.140.
   **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB)

6. The coefficient of the mineral bulk modulus in the calculation of the bulk modulus of the frame in the region. This is the $C_K$ term in equation 3.141. To make the frame bulk modulus independent of the mineral bulk modulus, this value should be set to zero.
   \[ K^{\alpha}_{frame} = C_K K^{\alpha}_{minerals} + \sum_{i=0}^{\alpha} C_{K,i}(P)\phi_{eff}^i \]  
   **UNITS:** dimensionless

7. The coefficient of the mineral shear modulus in the calculation of the shear modulus of the frame in the region. This is the $C_G$ term in equation 3.142. To make the frame shear modulus independent of the mineral shear modulus, this value should be set to zero.
   \[ G^{\alpha}_{frame} = C_G G^{\alpha}_{minerals} + \sum_{i=0}^{7} C_{G,i}(P)\phi_{eff}^i \]  
   **UNITS:** dimensionless
where $\alpha$ is defined in the previous item, $G_{\text{minerals}}$ is defined in the next item, and the $C_{G,i}(P)$ tables are specified using the keyword PEGTABx.

UNITs: dimensionless

8. The shear modulus of the mineral, $G_{\text{minerals}}$, in the region.

UNITs: bars (METRIC), psia (FIELD), atm (LAB)

9. The exponent of the mineral bulk/shear modulus in the calculation of the bulk/shear modulus of the frame in the region. This is the $\alpha$ term in equations 3.141 and 3.142. Typical values for this are +1 (so that the mineral term is simply added to the pore volume terms to generate the frame term) and -1 (so that a harmonic style addition is used). A value of zero is not allowed for this item.

DEFAULT: 1

UNITs: dimensionless

10. The exponent in the calculation of bulk modulus of fluid. This is the $\epsilon$ term in equation 3.143.

$$K_{\text{fluid}} = (K_{\text{liquid}} - K_g)(1 - S_g)^\epsilon + K_g$$

Eq. 3.143

where

- $K_{\text{fluid}}$ is the bulk modulus of fluid (mixture of oil, water and gas),
- $K_g$ is the bulk modulus of gas, $S_g$ is the gas saturation and
- $K_{\text{liquid}}$ is given by equation 3.144.

Note that this item will be ignored if item 11 is 0.

$$\frac{S_o + S_w}{K_{\text{liquid}}} = \frac{S_o}{K_o} + \frac{S_w}{K_w}$$

Eq. 3.144

DEFAULT: 1

UNITs: dimensionless

11. Switch between the two available methods for calculating the fluid bulk modulus. These methods are a generalized form of Wood’s law for the oil, water and gas as given by equation 3.145,

$$\frac{1}{K_{\text{fluid}}} = \frac{S_o}{K_o} + \frac{S_w}{K_w} + \frac{S_g}{K_g}$$

Eq. 3.145

and the standard Wood’s law for the oil and water with Brie et al (SPE 30595) used for the addition of the gas term, as given in equation 3.143. This switch is an integer value. It can only take the values 0 and 1.

- A value of 0 indicates that equation 3.145 is to be used in the region
- A value of 1 indicates that equations 3.143 and 3.144 are to be used.

DEFAULT: 0
Example

The following example shows the PECOEFS for a model which has FIELD units and has 4 PENUM regions. The number of PENUM regions is specified by item 1 of the PEDIMS keyword in the RUNSPEC section.

In this example, the third and fourth rows have been left blank, so the data for these rows is copied from the second row.

|
|---|
|PECOEFS  |
|3.0E4 201.0 165.0 -2.8E6 2.2E6 0.5 1.0 0.9E6 1 2 1 / |
|3.0E4 210.0 168.0 -2.7E6 2.2E6 0.5 1.0 1.1E6 1 2 1 / |
|/ |
**PEDERSEN**  Use Pedersen et al. viscosity calculation

This keyword enables the viscosities to be calculated using the "The Pedersen method" (in the *ECLIPSE Technical Description*) rather than the default method, which is the "The Lorentz-Bray-Clark method" (in the *ECLIPSE Technical Description*).

There is no data associated with this keyword.

*Note:* The coefficients used in the mixture molecular weight calculation, which forms part of this viscosity correlation, may be modified by multiplicative tuning parameters specified using either the `PEDTUNE` or the `PEDTUNER` keywords.

**Example**

```plaintext
PEDERSEN
```

---

**Keywords**

1636
Petro-elastic dimension data

The data consists of two items, describing the number of regions and the maximum table lengths. The tables are entered in the PROPS section by keywords PEKTABx and PEGTABx.

The data consists of two integers terminated by a slash (/).

1. The number of regions to be entered in the PENUM keyword.
   DEFAULT: 0

2. The maximum number of pressure points in each table for keywords PEKTABx and PEGTABx.
   DEFAULT: 0

Example

```
PEDIMS
3 8 /
```
PEDTUNE    Set non-default Pedersen viscosity tuning parameters

The keyword provides a mechanism for modifying the default tuning parameters used for the calculation of
the mixture molecular weight and the rotational coupling factors in the Pedersen et al. viscosity correlation.
It is only relevant if the Pedersen et al. viscosity calculations have been selected by including the
PEDERSEN keyword in the PROPS section; otherwise it will be ignored.

The data consist of six items. items 1 and 2 are for the tuning mixture molecular weight and items 3 to 6 are
for the rotational coupling factors.

1. Tuning parameter 1:
   DEFAULT:  1.0
   Minimum:   0.1
   Maximum:  10.0

2. Tuning parameter 2:
   DEFAULT:  1.0
   Minimum:   0.1
   Maximum:  10.0

3. Tuning parameter 3:
   DEFAULT:  1.847
   Minimum:  -100.0
   Maximum:  100.0

4. Tuning parameter 4:
   DEFAULT:  0.5173
   Minimum:  -1000.0
   Maximum:  1.5

5. Tuning parameter 5:
   DEFAULT:  7.378E-03
   Minimum:  -0.015
   Maximum:  0.1

6. Tuning parameter 6:
   DEFAULT:  3.1E-02
   Minimum:  -0.2
   Maximum:  1.1

The data must be terminated with a slash (/).
Using this keyword only one set of tuning parameters is required. Alternatively, if multiple equation of state regions have been specified, the PEDTUNER keyword may be used to specify these tuning parameters separately for each equation of state.

For the Pedersen viscosity correlation, the mixture molecular weight is given by:

\[
M_{w,\text{mix}} = c_1 \cdot 1.304 \times 10^{-4} \left( \frac{M_w}{\bar{M}_w} \right)^{2.303} \left( \frac{M_n}{\bar{M}_n} \right)^{2.303} + M_n
\]

where

\( \bar{M}_w \) and \( \bar{M}_n \) are the weight average and number average mole weights respectively, and \( c_1 \) and \( c_2 \) are the tuning parameters 1 and 2 respectively, specified using this keyword.

For the Pedersen viscosity correlation, the rotational coupling factors are given by:

\[
\alpha_{\text{mix}} = 1 + c_5 \cdot \left( \rho_{r,\text{mix}} \right)^{c_3} \cdot \left( M_{w,\text{mix}} \right)^{c_4}
\]

\[
\alpha_o = 1 + c_6 \cdot \left( \rho_{r,\text{mix}} \right)^{c_3}
\]

where

\( \rho_{r,\text{mix}} \) is the reduced density, and \( c_3, c_4, c_5 \) and \( c_6 \) are the tuning parameters 3, 4, 5 and 6 respectively, specified using this keyword.

**Example**

Select a single set of non-default tuning parameters for the entire reservoir model:

```
PEDTUNE
1.001 1.005 1.847 0.5173 7.378E-03 3.1E-02 /
```
**PEDTUNER**  Set non-default Pedersen viscosity tuning parameters for multiple EoS regions

The keyword provides a mechanism for modifying the default tuning parameters used for the calculation of the mixture molecular weight and the rotational coupling factors in the Pedersen et al. viscosity correlation. This is done separately for each equation of state (EoS) region, the number of which is specified using item 9 in **TABDIMS**. This keyword is only relevant if the Pedersen et al. viscosity calculations have been selected by including the **PEDERSEN** keyword in the **PROPS** section and will otherwise be ignored.

The data consists of six items followed by a slash (/) for each equation of state region. Items 1 and 2 are for tuning the mixture molecular weight. Items 3 to 6 are for the rotational coupling factors.

1. Tuning parameter 1:
   - DEFAULT: 1.0
   - Minimum: 0.1
   - Maximum: 10.0
2. Tuning parameter 2:
   - DEFAULT: 1.0
   - Minimum: 0.1
   - Maximum: 10.0
3. Tuning parameter 3:
   - DEFAULT: 1.847
   - Minimum: -100.0
   - Maximum: 100.0
4. Tuning parameter 4:
   - DEFAULT: 0.5173
   - Minimum: -1000.0
   - Maximum: 1.5
5. Tuning parameter 5:
   - DEFAULT: 7.378E-03
   - Minimum: -0.015
   - Maximum: 0.1
6. Tuning parameter 6:
   - DEFAULT: 3.1E-02
   - Minimum: -0.2
   - Maximum: 1.1

The data must be terminated with a slash (/).

For the Pedersen viscosity correlation, the mixture molecular weight is given by:
\[ M_{w,\text{mix}} = c_1 \cdot 1.304 \times 10^{-4} \left( M_w^{c_2^{2.303}} - M_n^{c_2^{2.303}} \right) + M_n \]

where

\( M_{w,\bar{c}_1} \) and \( M_{n,\bar{c}_1} \) are the weight average and number average mole weights respectively, and \( \bar{c}_1 \) and \( \bar{c}_2 \) are the tuning parameters 1 and 2 respectively, specified by this keyword.

For the Pedersen viscosity correlation, the rotational coupling factors are given by:

\[ \alpha_{\text{mix}} = 1 + c_5 \cdot (\rho_{r,\text{mix}})^{c_3} \cdot (M_{w,\text{mix}})^{c_4} \]

\[ \alpha_o = 1 + c_6 \cdot (\rho_{r,\text{mix}})^{c_3} \]

where

\( \rho_{r,\text{mix}} \) is the reduced density, and \( c_3, c_4, c_5 \) and \( c_6 \) are the tuning parameters 3, 4, 5 and 6 respectively, specified using this keyword.

If only one set of tuning parameters is required, the \texttt{PEDTUNE} keyword may be used to specify these tuning parameters for the entire reservoir.

**Example**

Select non-default tuning parameters for a model with two equation of state regions:

```
PEDTUNER
  1.001  1.005  1.847  0.5173  7.378E-03  3.1E-02 /
  1.002  1.006  1.84  0.5173  7.378E-03  3.15E-02 /
```

Keywords

**1641**
Pressure table for shear modulus of frame

These keywords specify the tables which contain the coefficients of a polynomial in effective porosity $\phi_{eff}$ to be used in the calculation of the shear modulus of the frame in the petro-elastic model. Each of these tables specifies a coefficient as a function of pressure $P$ for each PENUM region.

The general version of shear modulus of frame, $G_{frame}$ is

$$G_{frame}^\alpha = C_G G_{mineral}^\alpha + \sum_{i=0}^{7} C_{G,i}(P) \phi_{eff}^i$$

where

$C_G$, $G_{mineral}$ and $\alpha$ are the constants defined by keyword PECOEFS in the PROPS section,

$\phi_{eff}$ is the effective porosity and

the $C_{G,i}(P)$ terms are the following keywords recognized by ECLIPSE:

- PEGTAB0 Table of $C_{G,0}(P)$ against pressure $P$
- PEGTAB1 Table of $C_{G,1}(P)$ against pressure $P$
- PEGTAB2 Table of $C_{G,2}(P)$ against pressure $P$
- PEGTAB3 Table of $C_{G,3}(P)$ against pressure $P$
- PEGTAB4 Table of $C_{G,4}(P)$ against pressure $P$
- PEGTAB5 Table of $C_{G,5}(P)$ against pressure $P$
- PEGTAB6 Table of $C_{G,6}(P)$ against pressure $P$
- PEGTAB7 Table of $C_{G,7}(P)$ against pressure $P$

Each keyword consists of a number of tables. The number of tables is determined by item 1 of keyword PEDIMS in the RUNSPEC section, there being one table for each PENUM region. The number of entries in a single table should not exceed item 2 of keyword PEDIMS. Each table must be terminated by a slash (/).

If a keyword is not present, all its tables are assumed to be zero.

Each table consists of the following items of data.

1. The pressure $P$
   The pressure values - these must be monotonically increasing for each table.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB)

2. The value of $C_{G,i}(P)$
   UNITS: barsa$^i$ (METRIC), psia$^i$ (FIELD), atma$^i$ (LAB)

Empty tables (containing just their terminating slash) default to a copy of the previous table.
Example

If the information in the PEDIMS keyword specifies that there are 3 PENUM regions and at least 4 points in each table, then the PEGTAB7 keyword could be as shown:

```
PEGTAB7
  1800 1.0E+5
  2000 3.1E+5
  2100 2.8E+5
  2480 2.7E+5 /
  1500 0.8E+5
  2500 4.2E+5
  3000 2.9E+5 /
 /
```

In this example, the table for the third PENUM region is defaulted to be the same as that for the second region.
Pressure table for bulk modulus of frame

These keywords specify the tables which contain the coefficients of a polynomial in effective porosity $\phi_{\text{eff}}$ to be used in the calculation of the bulk modulus of the frame in the petro-elastic model. Each of these tables specifies a coefficient as a function of pressure $P$ for each PENUM region.

The general version of bulk modulus of frame, $K_{\text{frame}}$, is

$$K_{\text{frame}}^\alpha = C_K K_{\text{mineral}}^\alpha + \sum_{i=0}^{7} C_{K,i}(P) \phi_{\text{eff}}^i$$

Eq. 3.146

where

$K_{\text{mineral}}$ is the bulk modulus of mineral material making up rock,

the constants $C_K$ and $\alpha$ are defined in the keyword PECOEFS in the PROPS section,

$\phi_{\text{eff}}$ is the effective porosity and

the $C_{K,i}(P)$ terms are the following keywords recognized by ECLIPSE:

PEKTAB0 Table of $C_{K,0}(P)$ against pressure $P$

PEKTAB1 Table of $C_{K,1}(P)$ against pressure $P$

PEKTAB2 Table of $C_{K,2}(P)$ against pressure $P$

PEKTAB3 Table of $C_{K,3}(P)$ against pressure $P$

PEKTAB4 Table of $C_{K,4}(P)$ against pressure $P$

PEKTAB5 Table of $C_{K,5}(P)$ against pressure $P$

PEKTAB6 Table of $C_{K,6}(P)$ against pressure $P$

PEKTAB7 Table of $C_{K,7}(P)$ against pressure $P$

Each keyword consists of a number of tables. The number of tables is determined by item 1 of keyword PEDIMS in the RUNSPEC section, there being one table for each PENUM region. The number of entries in a single table should not exceed item 2 of keyword PEDIMS. Each table must be terminated by a slash (/).

If a keyword is not present, all its tables are assumed to be zero.

Each table consists of the following items of data.

1. The pressure $P$
   - The pressure value. These must monotonically increase for each table.
   - UNITS: barsa (METRIC), psia (FIELD), atma (LAB)
2. The value of $C_{K,i}(P)$
UNITs: barsa (METRIC), psia (FIELD), atma (LAB)

Empty tables (containing just their terminating slash) default to a copy of the previous table.

Example

If the information in the PEDIMS keyword specifies that there are 3 PENUM regions and at least 4 points in each table, then the PEKTAB4 keyword could be as shown:

```
PEKTAB4
1240 0.1E+6
2357 2.0E+6
2450 1.1E+6 /
1000 4.4E+6
2000 2.6E+6
2500 2.7E+6
3000 3.0E+6 /
/
```

In this example, the table for the third PENUM region is defaulted to be the same as that for the second region.
PENUM

Petro-elastic region numbers

The keyword should be followed by one integer for every grid block in the current input box specifying the region to which it belongs.

**CAUTION:** You must specify a value for each grid block in the model.

The region number should not be less than 1 nor greater than the first entry in the keyword PEDIMS. The data must be terminated by a slash (/).

The petro-elastic region number specifies which set of petro-elastic coefficients and frame bulk/shear modulus functions (input using PECOEFS, PEKTABx and PEGTABx in the PROPS section) should be used to calculate the petro-elastic properties in each grid block.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

See also the petro-elastic function keywords PEDIMS, PECOEFS, PEKTABx and PEGTABx in the PROPS section.

**Example**

With NDIVIX=10, NDIVIY=10, NDIVIZ=3 specified in RUNSPEC keyword DIMENS and no input BOX set, the following example would specify that the top layer is in petro-elastic region 1, the middle layer is in region 2, and the third layer is in region 3:

```
PENUM
100*1 100*2 100*3 /
```
PERMAVE

Permeability averages for transmissibility

This keyword may optionally be used to modify the way the adjacent grid block permeabilities are averaged when ECLIPSE calculates the interblock transmissibilities.

The keyword should be followed by three real numbers, to specify the exponents used in the power law expression for averaging block permeabilities in each direction. These replace the default weighted harmonic averages of block permeabilities employed when calculating interblock transmissibilities. The three numbers represent the exponents for the permeability averages in the I, J and K directions respectively. The data should be terminated by a slash (/).

Each exponent is defaulted to -1.0, which reproduces the usual weighted harmonic averages of block permeabilities. A value of 1.0 produces a weighted arithmetic average, while a value of 0.0 produces a weighted geometric average. If the exponent is less than -10.0 (or greater than 10.0), then the minimum (or maximum) block permeability respectively is used as the average.

If the LINKPERM keyword is specified as well as PERMAVE, then the interface permeability implied by LINKPERM overrides any average indicated by PERMAVE.

The forms taken by the various options are described in "Transmissibility Calculations" in the ECLIPSE Technical Description.

Example

The default transmissibilities are used in the X direction, while Y direction permeabilities use a geometric average and arithmetic averaging is used for the Z direction permeabilities.

```plaintext
PERMAVE
1* 0.0 1.0 /
```
PERMMF

Permeability for matrix-fracture coupling

The PERMMF keyword is followed by a value for each matrix cell. $N_x \cdot N_y \cdot N_z / 2$ values are required in all, in normal grid order with the X-index changing fastest.

UNITS: mD (METRIC), mD (FIELD), mD (LAB), mD (PVT-M)

---

<table>
<thead>
<tr>
<th>Keywords</th>
</tr>
</thead>
<tbody>
<tr>
<td>ECLIPSE 100</td>
</tr>
<tr>
<td>ECLIPSE 300</td>
</tr>
<tr>
<td>SPECIAL</td>
</tr>
<tr>
<td>RUNSPEC</td>
</tr>
<tr>
<td>GRID</td>
</tr>
<tr>
<td>EDIT</td>
</tr>
<tr>
<td>PROPS</td>
</tr>
<tr>
<td>REGIONS</td>
</tr>
<tr>
<td>SOLUTION</td>
</tr>
<tr>
<td>SUMMARY</td>
</tr>
<tr>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>

---

1648
Specifies radial permeability values

This keyword specifies permeability values in the radial direction for a radial run.

The keyword should be followed by one non-negative real number for every grid block in the current input box specifying the R-direction permeability. The data must be terminated by a slash (/).

UNITS: mD (METRIC), mD (FIELD), mD (LAB), mD (PVT-M)

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta- and Z-axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

In ECLIPSE 100, every PERMR value in the top plane (K = 1) must be specified in one way or another by the end of the GRID section. Values in lower planes (K > 1) that are not specified default to the value in the plane above.

Example

```
-- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX
  5  16  3  8  1  1  /
PERMR
  12*100
  12*8
  12*2300
  12*100
  12*20
  12*38  /
```
PERMSTAB  Geomechanics permeability multiplier as a tabular function of rock stress

A permeability multiplier may be specified in table form as a function of maximum and minimum principal stress. This keyword is only valid if the geomechanics option is chosen (see GEOMECH in the RUNSPEC section).

The data comprises NGMYRG (first item in keyword GEODIMS in the RUNSPEC section) tables, each terminated by a slash (/), where NGMYRG is the number of geomechanics regions in the model.

The first record sets the values of the maximum principal stress, \( \sigma_1 \). These must be monotonically increasing. This record must end with a slash (/).

Record 1  \( \sigma_{1,1} \quad \sigma_{1,2} \quad \ldots \quad \sigma_{1,NGMPTC} \)

The next NGMPTR records set the values of the minimum principal stress, \( \sigma_3 \) followed by the table data for the permeability multipliers, \( PM \). A value of 1 for a multiplier means the phase mobilities are unchanged. Values of the minimum principal stress must be monotonically increasing; that is, subsequent records must have increasing \( \sigma_3 \).

Repeat counts may be used with the multipliers, but not with the maximum/minimum principal stresses.

Record 2  \( \sigma_{3,1} \quad PM_{1,1} \quad \ldots \quad PM_{1,NGMPTC} \)

Record 3  \( \sigma_{3,2} \quad PM_{2,1} \quad \ldots \quad PM_{2,NGMPTC} \)

\[ \vdots \quad \vdots \quad \vdots \]

Record NGMPTR  \( \sigma_{3,NGMPTR} \quad PM_{NGMPTR,1} \quad \ldots \quad PM_{NGMPTR,NGMPTC} \)

Records 2 through NGMPTR must end with a slash (/).

**Note:** If the permeability multiplier is only a function of the maximum principal stress, then set or default the number of rows, NGMPTR, to be 1. In this case, the first value in Record 2, \( \sigma_{3,1} \), is ignored. Similarly, if the multiplier is only a function of the minimum principal stress, then set or default the number of columns, NGMPTC, to be 1. Again, in this case the first value in Record 1, \( \sigma_{1,1} \), is ignored.

The set of records is ended by a blank record, containing only a slash.

The permeability multipliers are not checked for monotonicity. If input maximum and minimum principal stresses from the simulation model lie outside the range of the above table, constant slope extrapolation is used.

See also "Geomechanics" in the ECLIPSE Technical Description.

**Example**

With NGMYRG=1, NGMPTC=4, NGMPTR=3:
```plaintext
PERMSTAB

-- Maximum principal stresses
   5000  7000  9000  11000
-- Minimum principal stress, permeability multipliers
  5000  2*1  1.1  1.3
  6000  1  1.1  1.2  1.4
  8000  1.1  1.3  1.6  1.9
/
```
PERMTHT  Specifies azimuthal permeability values

This keyword specifies permeability values in the azimuthal direction for a radial run.

The keyword should be followed by one non-negative real number for every grid block in the current input box specifying the theta-direction permeability. The data must be terminated by a slash (/).

UNITs: mD (METRIC), mD (FIELD), mD (LAB), mD (PVT-M)

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta- and Z-axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

In ECLIPSE 100, every PERMTHT value in the top plane (K = 1) must be specified in one way or another by the end of the GRID section. Values in lower planes (K > 1) that are not specified default to the value in the plane above.

Example

```
-- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX
  5 16 3 8 1 1 /
PERMTHT
  12*100
  12*8
  12*2300
  12*100
  12*20
  12*38 /
```
PERMX

Specifies X-permeability values

This keyword specifies the permeability values in the X-direction.

The keyword should be followed by one real number for every grid block in the current input box specifying the X-direction permeability. Permeability values must be non-negative, unless the MPFA keyword is used. The data must be terminated by a slash (/).

UNITS: mD (METRIC), mD (FIELD), mD (LAB), mD (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

Note: This is not the permeability between a cell and its neighbor. The transmissibility between a cell and its neighbor involves the permeability values for both cells.

Every PERMX value in the top plane (K = 1) must be specified in one way or another by the end of the GRID section. Values in lower planes (K > 1) that are not specified default to the value in the plane above.

Example

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX      5 16 3 8 1 1 /
PERMX
  100 1500 10*60
  90 1500 10*60
  80 1500 10*65
  70 1500 10*70
  60 1500 10*75
  50 1500 10*75 /
```
PERMXY-ZZ  Specifies off-diagonal tensor permeability coefficients

The PERMXY, PERMYZ, PERMZX, PERMXX, PERMYY and PERMZZ keywords specify the components of the permeability tensor. The basis in which the tensor is defined is given by the MPFA keyword. The permeability tensor $K$ is assumed to be symmetric and is specified by its components $K_{11} = \text{PERMX}$, $K_{22} = \text{PERMY}$, $K_{33} = \text{PERMZ}$, $K_{12} = K_{21} = \text{PERMXY}$, $K_{23} = K_{32} = \text{PERMYZ}$ and $K_{31} = K_{13} = \text{PERMZX}$.

The diagonal terms are aliased: PERMXX to PERMX, PERMYY to PERMY and PERMZZ to PERMZ.

The keyword should be followed by one real number for every grid block in the current input box specifying the X-direction permeability. The data must be terminated by a slash (/).

**UNITS:** mD (METRIC), mD (FIELD), mD (LAB), mD (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

Keyword MPFA must be specified to enable the use of tensor permeabilities.

Any tensor values not entered are defaulted to zero (0.0).

**Example**

```
------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX      5  16  3  8  1  1  /
PERMXY
 100 1500 10*60
  90 1500 10*60
  80 1500 10*65
  70 1500 10*70
  60 1500 10*75
  50 1500 10*75 /
```
**PERMY**

**Specifies Y-permeability values**

This keyword specifies the permeability values in the Y-direction.

The keyword should be followed by one real number for every grid block in the current input box specifying the Y-direction permeability. Permeability values must be non-negative, unless the `MPFA` keyword is used. The data must be terminated by a slash (/).

**UNITS**: mD (METRIC), mD (FIELD), mD (LAB), mD (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

**Note:** This is not the permeability between a cell and its neighbor. The transmissibility between a cell and its neighbor involves the permeability values for both cells.

Every `PERMY` value in the top plane (K = 1) must be specified in one way or another by the end of the `GRID` section. Values in lower planes (K > 1) that are not specified default to the value in the plane above.

**Example**

```
--- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX  5 16 3 8 1 1 /
PERMY
  100 1500 10*60
  90 1500 10*60
  80 1500 10*65
  70 1500 10*70
  60 1500 10*75
  50 1500 10*75 /
```
**PERMZ**

**Specifies Z-permeability values**

This keyword specifies the permeability values in the Z-direction.

The keyword should be followed by one real number for every grid block in the current input box specifying the Z-direction permeability. Permeability values must be non-negative, unless the MPFA keyword is used. The data must be terminated by a slash (/).

**UNITS:** mD (METRIC), mD (FIELD), mD (LAB), mD (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

**Note:** This is not the permeability between a cell and its neighbor. The transmissibility between a cell and its neighbor involves the permeability values for both cells.

**Example**

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX      5  16  3  8  1  1  /
PERMZ    72*0  /
```
**Imports grid from a .GSG**

This keyword allows the grid geometry to be imported from either a Generic Simulation Grid (GSG) file. This type of file format has to be imported using Petrel.

ECLIPSE will read a .GSG file. The root extension must be provided.

ECLIPSE cannot generate a GRID/EGRID output file when the geometry is read from a GSG file with the PETGRID keyword. If an EGRID/GRID is required, export it from Petrel.

### .GSG files

The .GSG file can contain grid geometry for both global and local grids. Either corner point geometry with straight pillars or corner point geometry with curved or truncated pillars can be imported.

For any .GSG file containing LGRs, it is important to note that the PETOPTS keyword, together with the LGRSIMGD argument, must be set in the RUNSPEC section of the data file. This helps in the preprocessing of the LGRs within the simulators.

For straight pillar geometry, non-neighbor connections and transmissibilities can be determined by the simulators, but if values are present in the .GSG file then they will be used and no calculations will be performed. For curved or truncated pillar geometry they must be specified in the .GSG file. All other grid-cell based global and local properties for the GRID, EDIT, PROPS, REGIONS and SOLUTION sections can be imported. This includes any necessary DUALPORO and DUALPERM cell based properties, for instance the multiplier to be used in determining the matrix-fracture coupling transmissibilities.

---

**Note:** Where the transmissibilities are imported from the .GSG file rather than calculated, any keywords which would have been used in their initial calculation in the GRID section will have no effect. If transmissibility multipliers are required, they should be specified in the EDIT section.

Since the .GSG file is exported from Petrel, it is advisable that the geometry be viewed there. ECLIPSE will not output any kind of GRID file if the PETGRID keyword is present in the dataset. A warning message will be issued if GRID file output is requested when the PETGRID keyword is used.

### .OPF files

**Attention:** The PETGRID keyword was introduced in 2005A in order to import the .OPF file format which was introduced for the FrontSim simulator. From the 2008.1 release, this file format was no longer supported in ECLIPSE. From the 2012.1 release, the file format support is withdrawn. An alternate method must be chosen, for instance, using the .GSG format, importing GRID files and/or using ECLIPSE keywords.

---

### Specifying PETGRID keyword and data

The PETGRID keyword should be followed by a single data item specifying the complete name, including any suffix, of the .GSG file containing the required grid information.

The PETGRID keyword is incompatible with auto refinement. The grid should be altered inside Petrel in this case.

The data should be terminated by a slash (/).
**CAUTION:** This keyword is also not recommended for use with the vertical equilibrium option.

### Example

In this example, ECLIPSE reads the geometric grid data and cell properties from the GSG file called `MODEL1.GSG`.

```plaintext
PETGRID
  MODEL1.GSG /
```
PETOPTS Options for Petrel runs

The keyword should be followed by some or all of the following data items, which typically set options for simulator runs started by Petrel. The data should be terminated by a slash (/).

**Note:** Refer to Petrel documentation and release notes for additional information.

**CAUTION:** The LGRSIMGD, TRANPORV, NOFLTNNC and NOPINCHC arguments should never be edited by hand - they are written by Petrel to reflect the data contained in the .GSG file. Altering them by hand is likely to lead to invalid results.

PETOPTS data items

Simulator input options

**Note:** The following options affect simulator input. They must correspond with the contents of the geometry file in order for the simulator to function correctly. They should never be added or removed manually.

- **LGRSIMGD** The geometry file imported using the PETGRID keyword contains local grid refinements. This mnemonic is necessary for the simulators to read and process the local grid data during the pre-sweep of all the keywords in the entire data file. Without it, the simulators will fail to process the LGR data correctly, resulting in error messages and failure to simulate.

- **TRANPORV** The geometry file(s) imported from Petrel contains transmissibilities and pore volumes, and also non-neighbor connections due to faults and pinchouts. For ECLIPSE Thermal models, rock volumes, heat transmissibilities and thermal conductive non-neighbor connections will also be exported. The TRANPORV mnemonic will be output by Petrel if curved or listric pillars are exported in the .GSG file. Typically, this geometry is complex so Petrel is best placed to define the necessary non-neighbor connections, transmissibilities, pore volumes (and rock volumes). For simpler corner point geometries, this mnemonic is not needed as the simulators are able to calculate all the essential grid geometry and grid cell volume information needed. However, the more complex geometries (curved or listric pillars) may cause ECLIPSE to miscalculate these quantities. It is recommended that the RPTGRID and RPTGRIDL keywords are used with mnemonics ALLNNC, PORV, TRANX / TRANY / TRANZ, ROCKV and HEATTX / HEATTY / HEATTZ for further investigation.

Simulator output options

**Note:** The following options affect simulator output and can be specified by the user as required.

- **INITNNC** By default, the simulators will not output any kind of grid file if the PETGRID keyword is used. If the run contains non-neighbor connections, INITNNC enables the NNC index information to be defined in the .INIT file together with the TRANNNC information and the CORSNUM information. If you have chosen a combination of settings which mean that a
grid file is being output, then this information will be defined both in it and in the .INIT file.

TINYINIT  Output only the pore volume array and the transmissibility arrays (and the non-neighbor connection information if requested by INITNNC) to the .INIT file. If this option is not specified, the output to the .INIT file will remain unchanged.

EDITSUPP  Suppress warning messages that could be output due to non-neighbor connections not being found while reading EDITNNC or EDITNNCR data.

NEWTRAN  Force ECLIPSE 300 to use a corner-point type calculation of transmissibilities. This is equivalent to the ECLIPSE 100 NEWTRAN keyword, and is intended to allow calculation of X and Y fault transmissibilities in coarsened regions. In ECLIPSE 100, this mnemonic is permitted but has no effect.

ECLIPSE 300 only

OLDORNNC  Within the .INIT file, output the data for non-neighbor connections between global cells in the order used pre-2010.1. If this option is omitted, then the order matches that used in the restart and .FLUX files.

NOCOMPIN  Output values for all grid blocks to the .INIT file, not just active grid blocks.

Note: When this mnemonic is specified in ECLIPSE 100, the simulator will run in NOSIM mode with all cells treated as active cells in order to generate the required .INIT file. If other output files are required, save the uncompressed .INIT file, remove the NOCOMPIN mnemonic, and rerun.

ECLIPSE 100 only

NOSWATIN  Suppress the output of the SWATINIT array to the .INIT file if this array is not specified in the input data file. By default this array is written to the .INIT file as undefined values if not specified in the input data file.

DEFAULT: None of the above

Obsolete options

The following options are obsolete and should not be added to datasets:

- NOFLTNNC: Simulator fault non-neighbor connection calculations are always suppressed if the TRANPORV mnemonic is specified.
- NOPINCHC: Simulator pinchout non-neighbor connection calculations are always suppressed if the TRANPORV mnemonic is specified.

Example

Allow non-neighbor connection index information to be output to the .INIT file, and import transmissibilities and pore volumes from the .GSG file.

```
PETOPTS
  INITNNC TRANPORV /
```
**PICOND**

**Controls calculation of generalized pseudo-pressure option**

This keyword controls the calculation of the generalized pseudo-pressure option for modeling the effects of condensate dropout on the fluid mobilities at producing well connections (see "Special inflow equations" in the *ECLIPSE Technical Description*). The **PICOND** keyword is optional, as suitable defaults are applied in its absence.

The option is activated in gas condensate runs for individual wells by entering **GPP** in item 8 of keyword **WELSPECS**. In ECLIPSE 300 the option may also be activated for all wells by entering the **PSEUPRES** keyword.

The keyword should be followed by some or all of the following data items, terminated with a slash (/).

1. The maximum interval between pressure quadrature points below the dew point pressure, in the calculation of the generalized pseudo-pressure integral.

   **UNITS:** bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

   **DEFAULT:** 4 times atmospheric pressure

2. The maximum interval between pressure quadrature points above the dew point pressure, in the calculation of the generalized pseudo-pressure integral.

   **ECLIPSE 100**
   - If a zero or negative quantity is entered, the quadrature points are the same as the pressure points input in the **PVTG** table.
   - **UNITS:** bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   - **DEFAULT:** 0.0

   **ECLIPSE 300**
   - **DEFAULT:** 10 times atmospheric pressure

3. Damping coefficient, **PPDAMP**, for the blocking factor.

   This value must lie between 0.0 and 1.0. It provides a means of damping oscillations that may result from the explicit calculation of the blocking factor $\beta$ (which is calculated at the beginning of each timestep), by averaging it with its value from the previous timestep according to the formula

   \[
   \beta_{\text{used}} = \beta_{\text{calculated}} + (1 - \beta_{\text{PPDAMP}}) \beta_{\text{previous}}
   \]

   **DEFAULT:** 1.0

4. Allow Velocity Dependent Relative Permeability (VDRP) and Forchheimer effects in the pseudo-pressure calculation?

   - Either **YES** or **NO**

   If this data item is set to **YES**, both Capillary Number VDRP and Forchheimer effects are included in the pseudo-pressure calculation. The VDRP option must be activated with keyword **VELDEP**. Properties of the Capillary Number and Forchheimer models are set with keywords **VDKRO**, **VDKRG** and **VDKRGC**.

   **DEFAULT:** **NO**

5. Multiplicative factor, **PPBFAC**, to generate the lower bound in the construction of the generalized pseudo-pressure integral table.

   \[
   p_L = (\text{PPBFAC}) \times p_{\text{completion cell}}
   \]

---

**Keywords**

1661
This lower bound is the \( p_L \) base pressure described in "Generalized pseudo-pressure in ECLIPSE 300" in the ECLIPSE Technical Description. It should be chosen so that the required range of cell and well pressures will be covered by the pseudo-pressure integral.

This is a dimensionless value which must lie between 0.0 and 0.95.

DEFAULT: 0.0

**ECLIPSE 300 only**

6. Multiplicative factor, \( \text{PPAFAC} \), to generate the upper bound in the construction of the generalized pseudo-pressure integral table.

\[
p_U = (\text{PPAFAC}) \times p_{\text{completion cell}}
\]

This upper bound should be chosen so that the required range of cell and well pressures will be covered by the pseudo-pressure integral.

This is a dimensionless value which must lie between 1.05 and 2.0.

DEFAULT: 1.1

**ECLIPSE 300 only**

7. Dimensionless number, \( \text{PPSWLM} \), giving the fractional change in water saturation in the completion grid block necessary to force a recalculation of the generalized pseudo-pressure integral table.

DEFAULT: 0.1

**ECLIPSE 300 only**

8. Dimensionless number, \( \text{PPPSLM} \), giving the fractional decrease in saturation pressure in the completion grid block necessary to force a recalculation of the generalized pseudo-pressure integral table. An increase in saturation pressure always forces a recalculation.

DEFAULT: 0.1

**ECLIPSE 300 only**

9. Dimensionless number, \( \text{PPCMLM} \), giving the fractional change in composition necessary to force a recalculation of the generalized pseudo-pressure integral table. If any one component shows this fractional change then the table is recomputed.

DEFAULT: 0.01

**ECLIPSE 300 only**

10. Maximum time between recalculations of generalized pseudo-pressure integral table. If this is set to a negative value then no limit is imposed.

UNITs: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

DEFAULT: -1 (that is no limit)

**ECLIPSE 300 only**

11. Adaptive ordinate error. When calculating the generalized pseudo-pressure integral table, if this value is positive more ordinates are placed where the mobility is non-linear, and fewer ordinates are placed where the mobility is linear. This item determines how many ordinates are used. If the area defined by three ordinates is different from the area defined by the outer two by more than this fraction, an extra ordinate will be added between the first two ordinates. A reasonable value for this item would be between 0.001 and 0.05.

DEFAULT: -1 (that is use regular ordinates)

**ECLIPSE 300 only**

12. Adaptive ordinate minimum spacing. This is the minimum possible distance between any pair of ordinates when the adaptive ordinate spacing scheme is active.

UNITs: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

DEFAULT: 0.1 atm

End the data record with a slash (/).
Notes

ECLIPSE 300 only

• The lower and upper bound factors (items 5 and 6) can be used to reduce the size of the pressure range covered by the generalized pseudo-pressure integral. In conjunction with the intervals in items 1 and 2, this can be used to increase the accuracy of the integration without having to use too many quadrature points (which can be computationally expensive otherwise). Unfortunately there is no good rule for selecting these bounds so the defaults should be used unless they are leading to problems.

ECLIPSE 300 only

• The generalized pseudo-pressure integration table recalculation limits (items 7, 8, 9 and 10) may be used if certain anomalous behaviors are encountered in production. One example is if a well BHP drops sharply, but then recovers then an inappropriate table may have generated an anomalously high blocking factor. If this is suspected, setting any of items 7, 8, 9 or 10 to 0.0 will force a recalculation every step. This may fix the problem, but slow down the simulation considerably. Finding an intermediate value should remove any problem and retain much of the default performance. Unfortunately there is no good rule for selecting these limits so the defaults should be used unless they are leading to problems.

ECLIPSE 300 only

• On occasion a rapidly changing blocking factor may occur. Significant damping of such a blocking factor (that is small values of item 3) can lead to inaccurate results (for example, a kink in the well BHP). If this problem is encountered try setting the damping coefficient (item 3) to 1.0 (or as large a value as possible). If this still does not resolve the issue then setting the well to be a single-segment multisegment well (see item 62 in OPTIONS3) may fix the problem.

Example

```
PICOND
30.0  0.0  0.5 /```

Keywords

1663
PI scaling table dimension data

The data consists of two items, describing the number of tables of PI scaling factor versus maximum water cut, and the maximum number of entries in any table. The tables are entered in the SCHEDULE section by keyword **PIMULTAB**.

The data consists of two integers terminated by a slash (/).

1. The number of PI Scaling tables to be entered in the PIMULTAB keyword.
   - **DEFAULT**: 0
2. The maximum number of entries in any one of the PI scaling tables.
   - **DEFAULT**: 0

**Example**

```
PIMTDIMS
  2 8 /
```
PIMULTAB  Tables of well PI multiplier versus max water cut

This keyword may be used to input PI multiplier tables, which scale a well’s connection factors according to the maximum water cut it has so far achieved. The tables defined here are allocated to individual wells with the WPITAB keyword. The facility may be useful in approximating performance decline due to backflow of water.

The keyword is followed exactly by NTPIMT tables, each containing two columns and no more than NPPIMT rows of data. The dimensions NTPIMT and NPPIMT are entered in the RUNSPEC section keyword PIMTDIMS. Each table should be terminated with a slash (/).

1. Maximum water cut values.
   The values should increase monotonically down the column.
   UNITS: \text{sm}^3/\text{sm}^3\ (\text{METRIC}), \text{stb/stb}\ (\text{FIELD}), \text{scc/scc}\ (\text{LAB}), \text{sm}^3/\text{sm}^3\ (\text{PVT-M})

2. The corresponding values of the PI scaling factor.

Empty tables (containing just their terminating slash) default to a copy of the previous table. Extrapolation outside the range of a table returns the value of the nearest point in the table.

See also keywords WPITAB and WPIMULT (the latter may be used to scale well PIs by a constant factor).

Note: The PI scaling tables are not written to the Restart files. Accordingly they must be present in any restarted data set. Thus it is also possible to change the tables at a restart.

Example

With NTPIMT=2, NPPIMT must have been set ≥5:

```
PIMULTAB
   -- max   PI
   -- WCT   mult
   0.0 1.0
   0.1 0.95
   0.15 0.85
   0.4 0.55
   0.8 0.43 /
   0.0 1.0
   0.01 0.92
   0.04 0.85
   0.12 0.44
   0.4 0.32 /
```
PINCH

**Generates connections across pinched-out layers**

The keyword should be followed by some or all of the following items, which define the pinchout threshold thickness in the z-direction and the threshold gap between adjacent layers. The data record should be terminated by a slash (/).

1. **Pinchout threshold thickness.**
   
   **UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   
   **DEFAULT:** 0.001 (all unit sets)

2. **Option controlling generation of pinchouts when a minimum pore volume has been set using MINPV or MINPVV.**

   **GAP**  Allow NNCs across cells that are inactive due to MINPV even if the thickness exceeds the threshold.

   **NOGAP**  The threshold thickness is strictly observed whether or not the pinched-out cells are inactive due to MINPV.

   **DEFAULT:** GAP

**Note:** Regardless of the value of this argument, any cells with a pore volume less than 0.000001 (in all unit sets) are automatically considered to have a negligible pore volume available for flow and only allow a pinchout NNC across them if the threshold thickness entered as the first item is not exceeded.

**ECLIPSE 300 only**

**Note:** If the thermal option has been selected, the cell’s rock volume is also considered, and a cell is only considered to have a negligible pore volume available for flow if both the pore volume and rock volume are less than 0.000001 (in all unit sets).

3. **Maximum empty gap allowed between cells in adjacent grid layers for a non-zero transmissibility to exist between them.**

   **UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   
   **DEFAULT:** Infinity (1.0E20 in all unit sets)

4. **Option for calculating the pinchout transmissibility.**

   **TOPBOT**  The pinchout transmissibility is formed from the half-cell Z-direction transmissibilities of the active cells on either side of the pinched-out layer(s).

   **ALL**  The pinchout transmissibility is a harmonic average of the Z-direction transmissibilities of all the cells between the active cells on either side of the pinched-out layer(s). In ECLIPSE 300, this option is only available for Cartesian corner-point geometry.

   **DEFAULT:** TOPBOT

5. **Option for accounting for MULTZ through a pinched-out column, if item 4 has been set to TOPBOT. If item 4 has been set to ALL, any value specified for item 5 is ignored and TOP is used.**

   **TOP**  The transmissibility multiplier applied to the pinchout is the MULTZ for the active cell at the top of the pinchout.
ALL  The transmissibility multiplier applied to the pinchout is the minimum of the \texttt{MULTZ} values for the active cell at the top of the pinchout and all the inactive cells in the pinched-out column. This replaces \texttt{OPTIONS} keyword switch 17. In ECLIPSE 300, this option is only available for Cartesian corner-point geometry.

\begin{itemize}
\item \texttt{DEFAULT: TOP}
\end{itemize}

\textbf{Note:} In a run, the transmissibility multipliers for all cells not included in the loaded flux region are set to zero. Pinchouts will therefore not be created across any cells not in the active flux regions when the \texttt{ALL} option of item 5 is specified.

Pinched-out layers are represented in the grid as layers of inactive cells of zero thickness. The presence of a layer of inactive cells would normally prevent flow from crossing it, between the active cells immediately above and below.

If the keyword \texttt{PINCH} is entered, ECLIPSE will automatically generate non-neighbor connections between the active cells on either side of the pinched-out layer(s), allowing fluid to flow across it. Generally, the layer or layers of inactive cells are deemed to be pinched-out if their overall cell thickness is less than the specified threshold value.

When \texttt{PINCH} has been specified and if keyword \texttt{MINPV} or \texttt{MINPVV} has been used to set the minimum pore volume a cell must have to be active, then by default, a column of cells may be pinched-out if the pore volumes are less than the minimum value, even if the overall thickness exceeds the threshold thickness. If \texttt{MINPV} or \texttt{MINPVV} is used to switch off cells with small pore volumes (to enhance the convergence of the solver), these cells will not act as a barrier to vertical flow unless the second item equals \texttt{NOCAP} and/or the pore volume is less than 0.000001 (in all unit sets), and the overall thickness exceeds the threshold set in the first item. If the second item equals \texttt{GAP}, cells with pore volume of less than 0.000001 (in all unit sets) will only act as a barrier if that individual cell’s thickness exceeds the threshold set in the first item.

By default, the pinchout connection reflects the Z-transmissibility multiplier (\texttt{MULTZ}) specified in the upper of the two connected cells, item 5 can be set to modify this behavior such that the minimum value of \texttt{MULTZ} for all pinched out cells is used.

If the third item is defaulted or omitted, then a non-zero transmissibility exists between two adjacent active cells in the same vertical column, whatever the size of the empty gap between the cells, and when \texttt{MULTZ} is not zero for the upper of the two cells. Otherwise, if this gap exceeds the threshold empty gap, then the transmissibility is always set to zero. For a prospective pinchout connection, no NNC is formed if the sum of the empty gaps between all adjacent layers of inactive cells in the pinchout exceeds this threshold.

The \texttt{PINCH} keyword only affects pinchouts between layers, i.e. in the z-direction. To control generation of pinchouts within a layer, i.e. in the x or y direction, use \texttt{PINCHXY}.

\section*{Notes}
\begin{itemize}
\item Non-neighbor connections are not generated if the keyword \texttt{NONNC} has been entered in the \texttt{RUNSPEC} section.
\item Pinchout connections are only formed across inactive cells, whatever the value set for the threshold thickness; there is no connection across active cells.
\item The default method (\texttt{TOPBOT} in item 4) to calculate the transmissibility of a non-neighbor connection due to a pinchout uses a harmonic average of the half-cell Z-direction transmissibilities for the active cells on either side of the pinched-out layer(s). For block-centered geometry with \texttt{OLDTRAN} (see
"Transmissibility Calculations" in the ECLIPSE Technical Description) the pinchout transmissibility between top and bottom cells \( t \) and \( b \) is

\[
\frac{1}{\frac{1}{\text{Tr}_t} + \frac{1}{\text{Tr}_b}} - \text{MULTZ}_t
\]

where

\[
\text{Tr}_t = \text{CDARCY} \frac{2\text{PERM}_t \text{DX}_t \text{DY}_t}{\text{DZ}_t}
\]

is the half-cell transmissibility for the top cell, etc.

The case of corner-point geometry with NEWTRAN is analogous (see "Transmissibility Calculations" in the ECLIPSE Technical Description). If item 5 is set to ALL, then the multiplier used is the minimum value of \( \text{MULTZ} \) between all cells in the pinched-out column (that is \( \min (\text{MULTZ}_i) \) for \( i = t, \ldots, b-1 \)).

If the flag ALL is specified in item 4, then the pinchout transmissibility is calculated as a harmonic average

\[
\frac{n}{\sum \frac{1}{\text{TRANZ}_i}}
\]

where

the sum is over all cells \( i = t, \ldots, i = b-1 \) in the column between \( t \) and \( b \),

\( n \) is the number of cells being summed over, and

each transmissibility \( \text{TRANZ}_i \) is calculated as described in "Transmissibility Calculations" in the ECLIPSE Technical Description. In this case, item 5 is ignored.

• In corner-point geometry, the total thickness of inactive layers in the pinchout (compare item 1) is computed as the maximum difference in corner-point depths between the active cells on either side of the pinched-out layer(s), for each of the four coordinate lines surrounding the column.

In corner-point geometry, the thickness of the empty gap between two adjacent cells (see item 3) is taken as minimum difference in corner-point depths between the appropriate coordinate lines.

• When item 5 is set to ALL, note that the transmissibility multiplier for the cell at the top is not itself changed when calculating the multiplier to be used for the pinchout.

**Example**

```
PINCH
   --Threshold   Gap or
   --thickness   No Gap
   0.1    'GAP' /
```

**Keywords**

1668
**PINCHNUM**  Identifies pinchout regions

The **PINCHNUM** keyword identifies regions for calculation of pinchouts. The threshold thickness and options for calculating the pinchouts should be supplied for each **PINCHNUM** region using the **PINCHREG** keyword.

To use this keyword, **NRPINC** must be set greater than zero in item 3 of the **RUNSPEC** section keyword **GRIDOPTS**, to define the maximum number of **PINCHNUM** regions.

The **PINCHNUM** keyword should be followed by one integer for each grid cell, specifying the region number to which it belongs. The data must be terminated by a slash (/).

Any cells not explicitly assigned a value of **PINCHNUM** are taken to be in region 1.

See also keywords **PINCHREG** and **PINCH**.

In ECLIPSE 100:

- For use with the dual porosity option, only the matrix cells or the fracture cells need to be input. The cell values in the other porosity will be copied automatically. If the user wants to specify matrix cells and fracture cells separately, then item 117 in **OPTIONS** should be set greater than 1. This will also affect use of the **MULTNUM**, **FLUXNUM** or **OPERNUM** keywords.
- The **PINCHNUM** keyword cannot be used inside a **CARFIN/RADFIN...ENDFIN** bracket.

**Example**

To define 4 separate regions in a 4*4*3 field, with **NRPINC**=4:

```plaintext
PINCHNUM
1 1 2 2
1 1 2 2
3 3 4 4
3 3 4 4
1 1 2 2
1 1 2 2
3 3 4 4
3 3 4 4
1 1 2 2
1 1 2 2
3 3 4 4
3 3 4 4
/
```

Keywords

1669
PINCHOUT  Generates connections across pinched-out layers

This is a less flexible (and older) alternative to the PINCH keyword, which also generates connections across pinched-out layers. The difference is that the PINCH keyword allows you to specify a maximum threshold thickness for pinched-out cells, whereas if you use the PINCHOUT keyword the threshold thickness is fixed at 0.001 (in all unit sets).

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

See the PINCH keyword for more information on generating pinch-out connections.

When cells are inactive due to MINPV or MINPVV, pinchout connections are formed regardless of the threshold thickness, analogously to using the PINCH keyword.

The PINCHOUT keyword has no associated data.

Example

```
PINCHOUT
```
PINCHREG  Generates connections across pinched-out layers within regions

This keyword supplies data for calculating pinchouts in multiple regions. The regions should be identified using keyword PINCHNUM. If PINCHNUM is not specified, then PINCHREG applies to flux regions defined by FLUXNUM. If neither PINCHNUM nor FLUXNUM is specified then PINCHREG is ignored.

The PINCHREG keyword should be followed by NRPINC records (item 3 of keyword GRIDOPTS) when using PINCHNUM, or NTFREG records (item 4 of keyword REGDIMS) when using FLUXNUM, with each record terminated by a slash (/).

Each record refers to a separate PINCHNUM or FLUXNUM region and consists of some or all of the items, detailed below, which are the same as in the PINCH keyword. If the record is defaulted for a region, then no pinchouts are generated in that region. When calculating a pinchout between cells in different regions, the PINCHREG data is taken from the region containing the upper of the two cells.

1. Pinchout threshold thickness.
   
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   
   DEFAULT: 0.001 (all unit sets)

2. Option controlling generation of pinchouts when a minimum pore volume has been set using MINPV or MINPVV.

   GAP  Allow NNCs across cells that are inactive due to MINPV even if the thickness exceeds the threshold.

   NOGAP The threshold thickness is strictly observed whether or not the pinched-out cells are inactive due to MINPV.

   DEFAULT: GAP

3. Maximum empty gap allowed between cells in adjacent grid layers for a non-zero transmissibility to exist between them.

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

   DEFAULT: Infinity (1.0E20 in all unit sets)

4. Option for calculating the pinchout transmissibility.

   TOPBOT  The pinchout transmissibility is formed from the half-cell Z-direction transmissibilities of the active cells on either side of the pinched-out layer(s).

   ALL  The pinchout transmissibility is a harmonic average of the Z-direction transmissibilities of all the cells between the active cells on either side of the pinched-out layer(s).

   DEFAULT: TOPBOT

5. Option for accounting for MULTZ through a pinched-out column, if item 4 has been set to TOPBOT. This item is ignored if item 4 is set to ALL.

   TOP  The transmissibility multiplier applied to the pinchout is the MULTZ for the active cell at the top of the pinchout.
The transmissibility multiplier applied to the pinchout is the minimum of the MULTZ values for the active cell at the top of the pinchout and all the inactive cells in the pinched-out column. This replaces item 17 in OPTIONS.

DEFAULT: TOP

See the PINCH keyword for further information.

Example

With NRPINC=4

<table>
<thead>
<tr>
<th>PINCHREG</th>
<th>Gap or Empty</th>
<th>No Gap</th>
<th>Gap</th>
</tr>
</thead>
<tbody>
<tr>
<td>Threshold</td>
<td>--thickness</td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.1</td>
<td>1*</td>
<td>10.0</td>
<td>/</td>
</tr>
<tr>
<td>0.2</td>
<td>1*</td>
<td>20.0</td>
<td>/</td>
</tr>
<tr>
<td>1.0</td>
<td>'NOGAP'</td>
<td></td>
<td>/</td>
</tr>
</tbody>
</table>
Generates horizontal pinchout connections

The keyword should be followed by up to two real numbers, which define the X-direction and Y-direction pinchout threshold widths. The data record should be terminated by a slash (/).

1. Threshold width in the X or R direction for horizontal pinchouts
   
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   
   DEFAULT: 0.001 (all unit sets)

2. Threshold width in the Y or THETA direction for horizontal pinchouts
   
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   
   DEFAULT: 0.001 (all unit sets)

Columns which are pinched-out in the X (or Y) direction are represented in the grid as columns of inactive cells of zero width in the X (or Y) direction. The presence of such a column of inactive cells would normally prevent flow from crossing it in the X (or Y) direction, between the active cells on either side of it.

If the keyword PINCHXY is entered, ECLIPSE automatically generates X (or Y) direction horizontal non-neighbor connections between the active cells on either side of the pinched-out column(s), allowing fluid to flow through it in the X (or Y) direction. Columns are deemed to be pinched-out in the X (or Y) direction if their overall cell width in the X (or Y) direction is less than the specified threshold value.

The X (or Y) direction pinch-out connection reflects the X (or Y) direction transmissibility multiplier (MULTX or MULTY) specified in the left most (or backmost) of the two connected cells.

Notes

- Non-neighbor connections are not generated if the keyword NONNC has been entered in the RUNSPEC section.
- Pinchout connections are only formed across inactive cells, whatever the value set for the threshold thickness.

ECLIPSE 300 only

- Only horizontal pinchouts in the global grid will be generated. Currently, no local grid PINCHXY connections will be calculated.

Example

```
PINCHXY
1* 0.1 /
```
The PLMIXNUM keyword is used with the Polymer flooding option to define regions with different polymer characteristics. The region number entered with this keyword is used to determine the Todd-Longstaff mixing and/or the maximum polymer and salt concentrations used in the mixing parameter calculation of fluid component viscosity. See keywords PLMIXPAR and PLYMAX. The keyword should be followed by one integer for each grid cell in the matrix. The data entries should not exceed NPLMIX entered using keyword REGDIMS (item 10) in the RUNSPEC section. The data must be terminated by a slash (/).

If the keyword is not present all matrix cells are assigned a value of unity.

Example

A 4*4*3 grid with differing mixing characteristics in the middle layer of the reservoir:

```
PLMIXNUM
16*1
16*2
16*1 /
```
**Polymer Todd-Longstaff mixing parameter**

This keyword should only be used in runs which use the polymer model (keyword POLYMER).

The data consists of NPLMIX (see keyword REGDIMS item 10) records of Todd-Longstaff mixing data. Each record consists of an item of data terminated by a slash (/).

1. Todd-Longstaff mixing parameter for the viscosity calculation.

   The value of the mixing parameter for each polymer mixing region should be in the range 0.0 to 1.0 inclusive.

   The mixing parameter has no associated units.

*Note:* This keyword is entirely separate from TLMIXPAR and is required even if TLMIXPAR is used to set Todd-Longstaff mixing parameters for the MISCIBLE option.

**Example**

With NPLMIX=5:

```plaintext
--- RUNSPEC section:
REGDIMS
  9* 5 /
--- PROPS section:
PLMIXPAR
  0.35 /
  0.57 /
  0.49 /
  0.8  /
  1.0  /
```

**Keywords**

ECLIPSE Reference Manual

1675
**Polymer adsorption functions**

The data comprises **NTSFUN** (see item 1 in **RUNSPEC** keyword **TABDIMS**) tables of polymer adsorption functions, each terminated by a slash (/), describing the adsorption of polymer by the rock formation. Each table consists of 2 columns of data:

1. The local polymer concentration in the solution surrounding the rock. The values should increase monotonically down the column.
   
   **UNITS:** kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scc (LAB)

2. The corresponding saturated concentration of polymer adsorbed by the rock formation. This quantity is therefore the mass of adsorbed polymer per unit mass of rock.
   
   These values should increase monotonically down the column.
   
   **UNITS:** kg/kg (METRIC), lb/lb (FIELD), gm/gm (LAB)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than **NSSFUN** (item 3 in **RUNSPEC** keyword **TABDIMS**).

**CAUTION:** The first row in the adsorption tables should consist of a pair of zeroes.

See "Polymer flood model" in the *ECLIPSE Technical Description* for further information.

Polymer adsorption functions can also be specified using the **ADSORP** and **PLYADSS** keywords.

**Notes**

- Changes made in the **SCHEDULE** section will not be honored in a restart run unless the re-specification is also included in the restart data file. This may be done either by placing the new data for **PLYADS** in the **PROPS** section or by leaving the **SCHEDULE** section keywords in place and using the **SKIPREST** keyword.

- When the temperature option (keyword **TEMP**) is activated, it is possible to specify this keyword multiple times with different values for different temperatures by using the **PLYATEMP** keyword before each occurrence of **PLYADS** to specify the relevant temperature.

**Example**

With **NTSFUN=2** and **NSSFUN≥4**:

```plaintext
PLYADS
  .0000  .00000
  .0003  .00005
  .0005  .00007
  .0008  .00008
  .0010  .00008 /
  .0000  .00000
  .0010  .00010 /
```

*Keywords*
**PLYADSS  Polymer adsorption functions varying with salinity**

This keyword should only be used for polymer studies where the salt-sensitivity option has been activated (keyword **BRINE** in RUNSPEC).

The data comprises NTSFUN (see item 1 in RUNSPEC keyword **TABDIMS**) tables of polymer adsorption functions, each terminated by a slash (/), describing the adsorption of polymer by the rock formation.

Each table consists of up to NSSFUN elementary sets of records, each corresponding to a given solution polymer concentration. In the first record of each set, there are two items:

1. The local polymer concentration in the solution surrounding the rock.
   
   **UNITS:** kg/sm\(^3\) (METRIC), lb/stb (FIELD), gm/scc (LAB)

2. The corresponding saturated concentration of polymer adsorbed by the rock formation, for the solution polymer concentration specified by item 1, and the salt concentration given by item 1 in the corresponding **ADSALNOD** table. This quantity is therefore the mass of adsorbed polymer per unit mass of rock.
   
   **UNITS:** kg/kg (METRIC), lb/lb (FIELD), gm/gm (LAB)

In subsequent records of the set, for record N (where N runs from 2 to a value not greater than NSSFUN, which is item 3 in RUNSPEC keyword **TABDIMS**) there is a single data item, specifying the adsorbed polymer concentration at the solution polymer concentration specified in record 1, and the salt concentration specified by item N in the corresponding **ADSALNOD** table.

Each set of records for a solution polymer concentration value should be terminated by a slash (/) and each table should be terminated by an additional slash.

There must be the same number of entries in each column of a given table, which should not be less than 2 or greater than NSSFUN (item 3 in RUNSPEC keyword **TABDIMS**). The number of salinity concentration entries should also be the same for each solution polymer concentration record, and correspond to the number of entries in the corresponding **ADSALNOD** table.

**CAUTION:** The first solution polymer concentration value in each table, and all corresponding adsorbed concentrations at all salinities, should be zero.

**Note:** When the temperature option (keyword **TEMP**) is activated, it is possible to specify this keyword multiple times with different values for different temperatures by using the **PLYATEMP** keyword before each occurrence of **PLYADSS** to specify the relevant temperature.

See "Polymer flood model" in the **ECLIPSE Technical Description** for further information.

Polymer adsorption functions can also be specified using the **ADSORP** keyword.

**Example**

With NTSFUN=1 and NSSFUN≥4:

```plaintext
PLYADSS
  0.0000  0.0
  0.0
```

**Keywords**
<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.0002</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.0003</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.0004</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.0004/</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.0005</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.0006</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.0006/</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.0007</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.0007</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.0008/</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.0008</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.0008</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.0008</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.0008</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Keywords**

1678
### PLYATEMP  Polymer adsorption table temperatures

This keyword should only be used for polymer studies. It allows for temperature dependence of the polymer adsorption tables (keywords PLYADS or PLYADSS) in the PROPS section. If it is used, multiple adsorption tables can be used, with each table giving the adsorption data at one temperature. In this case, the keyword PLYATEMP is used to give the temperature value of the adsorption table which follows it.

Adsorption data for intermediate temperatures is calculated by interpolation between the tables. Adsorption data for temperatures outside the range specified by the PLYATEMP keywords is defined using the adsorption values for the maximum or minimum PLYATEMP temperature value, as appropriate.

The data for each occurrence of this keyword consists of a series of temperature values, one for each SATNUM region used, which give the region temperatures for the following polymer adsorption table:

1. **Temperature.**

   **UNITS:** °C (METRIC), °F (FIELD), °C (LAB)

   The temperatures should be followed by a slash to terminate the record.

   The sequence of PLYATEMP/PLYADS or PLYATEMP/PLYADSS keyword pairs should be in order of increasing temperature.

   A maximum number of NTSFUN (see TABDIMS) temperatures can be provided in each instance of PLYATEMP.

   A maximum number of NSSFUN (see TABDIMS) temperatures per SATNUM region can be specified using PLYATEMP and hence a maximum number of NSSFUN instances of PLYATEMP can be provided.

   If PLYATEMP is not used, only one PLYADS or PLYADSS table can be provided and the adsorption data is then independent of temperature.

See “Polymer flood model” in the ECLIPSE Technical Description for further information.

### Example

```plaintext
PLYATEMP
  .50 /
PLYADS
  .0000 .00000
  .0003 .00005
  .0005 .00007
  .0008 .00008
  .0010 .00008 /
  .0000 .00000
  .0010 .00010 /
PLYATEMP
  200 /
PLYADS
  .0000 .0000
  .0003 .0005
  .0005 .0007
  .0008 .0008
  .0010 .0008 /
  .0000 .0000
  .0010 .00010 /
```

---

**Keywords**

**1679**
PLYCAMAX  Specifies maximum polymer adsorption values for the Polymer Flood Model

This keyword is optional, and is used to customize data for the Polymer Flood Model on a cell-by-cell basis. See "Polymer flood model" in the ECLIPSE Technical Description for further information.

The keyword should be followed by one real number for each cell in the current input box. The values denote the maximum polymer adsorption value that is to be used in the calculation of the resistance factor for the aqueous phase in the Polymer Flood Model. The data must be terminated by a slash (/).

Note that a zero value should not be used even if a zero adsorption isotherm has been defined.

Any cells which do not have an individual value set using this keyword will use the value for their rock type which is set in item 5 of the PLYROCK keyword.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0.00012). Note that spaces must not be inserted on either side of the asterisk (*).

The keyword can be used in conjunction with the EQUALS, ADD and MULTIPLY keywords, and the OPERATE and OPERATER keywords.

Note: Only cells which have had a valid individual PLYCAMAX value set will be affected by these data manipulation keywords. Cells which will use a value from PLYROCK will be unaffected.

UNITS: kg/kg (METRIC), lb/lb (FIELD), gm/gm (LAB), kg/kg (PVT-M)

See also keywords PLYKRRF and PLYRMDEN.

Example

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 4 1 4 1 1 /
PLYCAMAX
8*0.00012 8*0.00014 /
```

Keywords
Polymer thermal degradation half life

This keyword describes the thermal degradation of polymer due to temperature. The data comprises NTPVT (see item 2 in RUNSPEC keyword TABDIMS) tables of polymer thermal degradation functions, each terminated by a slash (/). Each table consists of the following columns of data:

1. The local temperature in the solution.
   The values should increase monotonically down the column.
   UNITS: °C (METRIC), °F (FIELD), °C (LAB)
2. The corresponding half-life of the polymer.
   UNITS: day (METRIC), day (FIELD), hr (LAB)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (item 4 in RUNSPEC keyword TABDIMS).

Note: Changes made in the SCHEDULE section will not be honored in a restart run unless the re-specification is also included in the restart data file. This may be done either by placing the new data for PLYDHFLF in the PROPS section or by leaving the SCHEDULE section keywords in place and using the SKIPREST keyword.

See the "Polymer flood model" in the ECLIPSE Technical Description for further information.

Example

With NTPVT=2 and NPPVT≥5:

```
PLYDHFLF
  0.0  365.0
  50.0 240.0
 100.0 180.0
 150.0 150.0
 200.0  90.0 /
  0.0  365.0
 100.0  90.0 /
```
Effective salinity coefficient for polymer

This keyword may only be used when the multi-component brine model (keyword `BRINE` in conjunction with keyword `ECLMC` in the `RUNSPEC` section) is active in conjunction with the polymer option (keyword `POLYMER` in the `RUNSPEC` section).

The `PLYESAL` keyword activates the effective salinity calculation for polymer adsorption and defines the parameter, $\alpha_p$, in the expression for the effective salinity

\[
C_{SEP} = C_- + (\alpha_p - 1)C_{++}
\]

where

- $C_{SEP}$ denotes the effective salinity for the polymer
- $C_-$ denotes the anion concentration
- $C_{++}$ denotes the divalent cation concentration
- $\alpha_p$ denotes the effective salinity parameter

See "Brine Tracking" in the *ECLIPSE Technical Description* for details of this option.

The keyword should be followed NTSFUN records of data (see item 1 in keyword `TABDIMS`), each containing a single non-negative real number. Each record must be terminated by a slash (/).

**Example**

For NTSFUN=2:

```
PLYESAL
  -- alpha
  10 /
  15 /
```
**PLYKRRF**

**Specifies residual resistance factors for the Polymer Flood Model**

This keyword is optional, and is used to customize data for the polymer flood model on a cell-by-cell basis. See "Polymer flood model" in the *ECLIPSE Technical Description* for further information.

The keyword should be followed by one real number for each cell in the current input box. The values denote the residual resistance factor for use in the Polymer Flood Model. The data must be terminated by a slash (/).

Values must be greater than or equal to 1 and represent the decrease in the rock permeability to the aqueous phase when the maximum amount of polymer has been adsorbed.

Any cells which do not have an individual value set using this keyword will use the value for their rock type which is set in item 2 of the **PLYROCK** keyword.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*1.22). Note that spaces must not be inserted on either side of the asterisk (*).

The keyword can be used in conjunction with the **EQUALS**, **ADD** and **MULTIPLY** keywords, and the **OPERATE** and **OPERATER** keywords.

**Note:** Only cells which have had a valid individual PLYKRRF value set will be affected by these data manipulation keywords. Cells which will use a value from PLYROCK will be unaffected.

See also keywords **PLYRMDEN** and **PLYCAMAX**.

**Example**

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX     1   4   1   4   1   1   /
PLYKRRF
8*1.75 8*1.95 /
```
PLYMAX  

Polymer/salt concentrations for mixing calculations

The data comprises NPLMIX (see item 10 in RUNSPEC keyword REGDIMS) rows of maximum polymer and salt concentrations that are to be used in the mixing parameter calculation of the fluid component viscosities. Each table consists of the following items of data:

1. The value of the polymer concentration in the solution which is to be used in the calculation of the maximum polymer fluid component viscosity.
   
   UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scce (LAB), kg/sm$^3$ (PVT-M)

2. The value of the salt concentration in the solution which is to be used in the calculation of the maximum polymer fluid component viscosity. This item is ignored if the salt-sensitivity keyword BRINE is not specified in the RUNSPEC section.
   
   UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scce (LAB), kg/sm$^3$ (PVT-M)

Note: Changes made in the SCHEDULE section will not be honored in a restart run unless the re-specification is also included in the restart data file. This may be done either by placing the new data for PLYMAX in the PROPS section or by leaving the SCHEDULE section keywords in place and using the SKIPREST keyword.

See "Polymer flood model" in the ECLIPSE Technical Description for further information.

Example

With NPLMIX=2:

```
PLYMAX
 .0100 .0500 /
 .0050 .0300 /
```
**Polymer options**

This keyword sets optional parameters for the ECLIPSE 300 polymer model. See the "Polymer flood model" in the *ECLIPSE Technical Description* for further information.

1. The minimum water saturation for polymer effects.
   
   **DEFAULT:** 1e-6
   
   For cells with small water saturations, a small amount of polymer will give a high concentration of polymer in the water phase, which could have a large effect on the water phase viscosity. To prevent numerical errors effecting the solution, polymer concentrations are disregarded for cells where the minimum water saturation is below a threshold given by this item.

**Example**

```
PLOYOPTS
  2.e-6 /
```
PLYRMDEN  Specifies mass densities for the Polymer Flood Model

This keyword is optional, and is used to customize data for the polymer flood model on a cell-by-cell basis. See "Polymer flood model" in the ECLIPSE Technical Description for further information.

The keyword should be followed by one real number for each cell in the current input box. The values denote the mass density of this rock type at reservoir conditions, for use in the calculation of the polymer loss due to adsorption in the Polymer Flood Model. The data must be terminated by a slash (/).

Any cells which do not have an individual value set using this keyword will use the value for their rock type which is set in item 3 of the PLYROCK keyword.

If the surfactant model is active, then the mass density of the rock supplied in the PLYRMDEN or PLYROCK keyword is used, and the value supplied in SURFROCK is ignored.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*1880). Note that spaces must not be inserted on either side of the asterisk (*).

The keyword can be used in conjunction with the EQUALS, ADD and MULTIPLY keywords, and the OPERATE and OPERATER keywords.

Note: Only cells which have had a valid individual PLYRMDEN value set will be affected by these data manipulation keywords. Cells which will use a value from PLYROCK will be unaffected.

UNITs: kg/rm³ (METRIC), lb/rb (FIELD), gm/rcc (LAB), kg/rm³ (PVT-M)

See also keywords PLYKRRF and PLYCAMAX.

Example

```
BOX  1 4 1 4 1 1 /
PLYRMDEN  8*1880 8*2010 /
```
**PLYROCK**

**Specifies the polymer-rock properties**

The data comprises NTSFUN (see item 1 in RUNSPEC keyword **TABDIMS**) rows specifying the rock properties which are required for the Polymer Flood Model. Each table should be terminated by a slash (/) and requires the following items of data:

1. The dead pore space for this rock type.
   
   The value specified should be greater than or equal to 0, less than the maximum water saturation and less than unity.

2. The residual resistance factor for this rock type.
   
   This quantity must be greater than or equal to 1 and represents the decrease in the rock permeability to the aqueous phase when the maximum amount of polymer has been adsorbed.

3. The mass density of this rock type at reservoir conditions.
   
   This quantity is used in the calculation of the polymer loss due to adsorption.
   
   If the surfactant model is active, then the mass density of the rock supplied in the PLYROCK keyword is used, and the value supplied in SURFROCK is ignored.
   
   **UNITS:** kg/rm$^3$ (**METRIC**), lb/rb (**FIELD**), gm/rcc (**LAB**), kg/rm$^3$ (**PVT-M**)

4. The adsorption index to be used for this rock type.
   
   Possible values for the adsorption index are 1 or 2. If a value of 1 is selected, then the polymer adsorption isotherm is retraced whenever the local polymer concentration in the solution decreases. If a value of 2 is selected, then no polymer desorption may occur.
   
   **DEFAULT:** 1

5. The maximum polymer adsorption value that is to be used in the calculation of the resistance factor for the aqueous phase.
   
   Note that a zero value should not be used even if a zero adsorption isotherm has been defined.
   
   **UNITS:** kg/kg (**METRIC**), lb/lb (**FIELD**), gm/gm (**LAB**), kg/kg (**PVT-M**)

See "Polymer flood model" in the ECLIPSE Technical Description for further information.

**Note:** When non-zero initial polymer concentrations are specified via the **SPOLY** keyword, the concentration specified in that keyword will be honored and extra adsorbed polymer is added to the rock so that it is in equilibrium with the input fluid concentration.

**Related keywords**

Polymer-rock properties can be modified in SCHEDULE using the **PLYROCKM** keyword.

**Example**

With NTSFUN=2 specified in keyword **TABDIMS**:

```
PLYROCK
0.13 1.75 1880 2 0.00012 /
0.16 1.95 2010 1 0.00014 /
```
PLYROCKM  Specifies modifiers for polymer-rock properties

The data comprises NTSFUN (see item 1 in RUNSPEC keyword TABDIMS) rows specifying modifiers for the rock properties which are required for the polymer flood model (see PLYROCK for a more detailed description of each of the properties).

1. The dead pore space for this rock type.
   The value specified should be greater than or equal to 0, less than the maximum water saturation and less than unity. This replaces the previous value entered using PLYROCK or PLYROCKM and is an absolute value rather than a multiplier.
   DEFAULT: Previous value entered using PLYROCK or PLYROCKM.

2. The residual resistance factor for this rock type.
   Acts as a multiplier on the values entered using PLYROCK or PLYKRRF. Multipliers are not cumulative.
   DEFAULT: 1

3. The mass density of this rock type at reservoir conditions.
   Acts as a multiplier on the values entered using PLYROCK or PLYRMDEN. Multipliers are not cumulative.
   DEFAULT: 1

4. The adsorption index to be used for this rock type.
   This replaces the previous value entered using PLYROCK or PLYROCKM.
   DEFAULT: Previous value entered using PLYROCK or PLYROCKM.

5. The maximum polymer adsorption value that is to be used in the calculation of the resistance factor for the aqueous phase.
   Acts as a multiplier on the values entered using PLYROCK or PLYCAMAX. Multipliers are not cumulative.
   DEFAULT: 1

Note: Changes made in the SCHEDULE section will not be honored in a restart run unless the re-specification is also included in the restart data file. This may be done either by placing the new data for PLYROCK in the PROPS section or by leaving the SCHEDULE section keywords in place and using the SKIPREST keyword.

Example

With NTSFUN=2:

```
PLYROCKM
  0.13 1.15 0.9 2 1.5 /
  0.16 1.0  1.0 1 1.2 /
```

Keywords
This keyword is used to activate the polymer shear thinning/thickening option and to supply the shear effect data. The keyword is optional, and if omitted no shear effect calculations are performed.

For shear thinning, a positive feedback effect is possible whereby increased flow speed leads to reduced viscosity which in turn produces more increase in flow speed. If the shear thinning multiplier is too small, this will produce a runaway condition producing convergence problems. An alternative option for the input of shear-thinning multipliers is available via the PLYSHLOG keyword, which may reduce this effect by controlling the logarithmic derivative of the multiplier. This alternative may be more appropriate if the shear-thinning data shows log-log behavior. The PLYSHEAR multiplier can also be controlled as described in "Notes".

The data comprises NTPVT (see item 2 in RUNSPEC keyword TABDIMS) tables of polymer shear thinning/thickening data, each terminated by a slash (/). The data is supplied as a shear multiplier that is applied in the update of the effective water and polymer viscosities and thus in the update of the water phase and polymer flows. This multiplier is tabulated as a function of water phase/polymer flow velocity. Each table consists of 2 columns of data:

1. The water phase/polymer flow velocity.
   By default this velocity is calculated taking into account the mobile water saturation. The pre-2014.1 velocity calculation, which omits the mobile water saturation, may be activated by setting OPTIONS keyword item 228 to 1.
   The values should increase monotonically down the column. The first entry should be 0.0.
   UNITS: m/day (METRIC), ft/day (FIELD), cm/hr (LAB)

2. The corresponding factor applied to the effective water and polymer viscosities (entered using PVTW with PLYVISC, PLYVISCS, PLYVISCT or PLYVSCST) due to shear thinning/thickening of the polymer.
   Usually the first entry would be 1.0.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see the RUNSPEC keyword TABDIMS).

Note: Changes made in the SCHEDULE section will not be honored in a restart run unless the re-specification is also included in the restart data file. This may be done either by placing the new data for PLYSHEAR in the PROPS section or by leaving the SCHEDULE section keywords in place and using the SKIPREST keyword.

The resulting shear viscosities of the water and polymer, at a given polymer concentration and flow velocity, are:

\[
\mu_{w,sh} = \mu_{w,eff}^* Z_{w}^* \\
\mu_{p,sh} = \mu_{p,eff}^* Z_{p}^*
\]

where each viscosity multiplier is calculated in the form

\[
Z = \frac{1 + (P - 1)M}{P}
\]

and
\( \mu_{w,sh} \) is the shear water viscosity

\( \mu_{p,sh} \) is the shear polymer viscosity

\( \mu_{w,eff} \) is the effective water viscosity

\( \mu_{p,eff} \) is the effective polymer viscosity

\( P \) is the viscosity multiplier assuming no shear effect (entered using the PLYVISC, PLYVISCS, PLYVISCT, or PLYVSCST keywords).

\( M \) is the shear effect multiplier supplied in the PLYSHEAR keyword.

Note that for \( M = 1 \), or no shear effect, we recover the effective component viscosities, and for \( M = 0 \), or maximum shear thinning, the shear viscosities are

\[
\mu_{w,sh} = \frac{\mu_{w,eff}}{P}, \quad \mu_{p,sh} = \frac{\mu_{p,eff}}{P}
\]

which corresponds to the minimum viscosity that can be obtained. If the polymer concentration is zero (\( P=1 \)) we recover the effective component viscosities again, which, in that particular case equal the pure water viscosity.

**Notes**

- If the shear effect is not significant, the PLYSHEAR keyword should be omitted, as the calculation of the velocity-dependent viscosity increases the run time.

- The positive feedback effect noted above for shear thinning will occur for \( P \) significantly larger than 1.0 if the gradient of the PLYSHEAR data, \( \frac{d(\log(M))}{d(\log(v))} \times -1 \). When \( P \) is near 1.0 (on a cell-by-cell or well/well-segment basis) this condition may be relaxed but should be monitored on a case-by-case basis. The simulator will warn if the data in PLYSHEAR violates this condition. It is recommended to keep the PLYSHEAR data within this bounding condition.

- In versions of ECLIPSE before 2012.1, shear thinning only was allowed and multiplier values in column 2 were required to decrease monotonically down the column. This has now been relaxed to allow shear thickening; however, in some cases, convergence problems may result. Currently, there is one case in which the cause of these problems is known. If a well is set not to allow crossflow (WELSPECS item 10) then oscillation in the temporary closure pattern of well connections due to a backflow condition can cause the well not to converge. Note that crossflow is allowed now for polymer, and it is preferred to use crossflow as this is the physical condition. If it is required that crossflow is disallowed, then shear effects can exacerbate non-convergence. In this case consider using item 198 in OPTIONS which closes back-flowing connections in the event of non-convergence (but not for multisegment wells).

See the "Polymer flood model" in the ECLIPSE Technical Description for further information.

**Example**

With NTPVT=1 and NPPVT>=4 specified in keyword TABDIMS:

```
PLYSHEAR
    0.0  1.0
```
<table>
<thead>
<tr>
<th>Label</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
<td>0.8</td>
</tr>
<tr>
<td>3.0</td>
<td>0.75</td>
</tr>
<tr>
<td>6.0</td>
<td>0.7</td>
</tr>
<tr>
<td>10.0</td>
<td>0.68</td>
</tr>
</tbody>
</table>
PLYSHLOG  Polymer shear thinning/thickening logarithmic model

This keyword is used to activate the alternative (logarithm-based) polymer shear thinning/thickening option and to supply the required shear effect data. The keyword is optional and, if omitted, no shear effect calculations are performed. This keyword therefore activates a variant of the polymer shear thinning/thickening model activated by keyword PLYSHEAR.

The data comprises NTPVT (see item 2 in RUNSPEC keyword TABDIMS) tables of polymer shear thinning/thickening data. Each table consists of two records, each terminated by a slash (/). The data is supplied as a shear effect multiplier, tabulated as a function of water phase/polymer flow velocity, for a specified set of reference conditions. Note that if keyword SHRATE is specified then the multipliers are instead tabulated as a function of water/polymer flow shear rate.

Record 1

Three items of data:

1. The reference polymer concentration that is associated with the shear data provided.
   
   UNITS: kg/sm³ (METRIC), lb/stb (FIELD), gm/scce (LAB)
   
   Note that this item cannot be defaulted and must take a positive value.

2. The reference salinity, if appropriate, that is associated with the shear data provided.
   
   UNITS: kg/sm³ (METRIC), lb/stb (FIELD), gm/scce (LAB)
   
   Note that this item must be specified if either of the PLYVISCS or PLYVSCST keywords are used, and should otherwise remain defaulted.

3. The reference temperature, if appropriate, that is associated with the shear data provided.
   
   UNITS: °C (METRIC), °F (FIELD), °C (LAB)
   
   Note that this item must be specified if either of the PLYVISCT or PLYVSCST keywords are used, and should otherwise remain defaulted.

Record 2

A table consisting of two columns of data that correspond to the reference conditions specified in Record 1.

1. The water phase/polymer flow velocity or shear rate (shear rate if SHRATE is included), X.
   
   By default this velocity is calculated taking into account the mobile water saturation. The pre-2014.1 velocity calculation, which omits the mobile water saturation, may be activated by setting OPTIONS keyword item 228 to 1.
   
   The values must increase monotonically down the column. The first entry $X_1 = \varepsilon$ must be a small positive value in the range $0 < \varepsilon \leq 10^{-6}$.
   
   UNITS (velocity): m/day (METRIC), ft/day (FIELD), cm/hr (LAB).
   
   UNITS (shear rate): 1/day (METRIC), 1/day (FIELD), 1/hr (LAB).

2. The corresponding shear effect multiplier, $Z_{ref}$.
These values must be positive and the first multiplier, which therefore corresponds to the minimal velocity (or shear rate), must be equal to 1.0.

UNITs: Dimensionless.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see the RUNSPEC keyword TABDIMS).

**Note:** Changes made in the SCHEDULE section will not be honored in a restart run unless the re-specification is also included in the restart data file. This may be done either by placing the new data for PLYSHLOG in the PROPS section or by leaving the SCHEDULE section keywords in place and using the SKIPREST keyword.

ECLIPSE takes each velocity (or shear rate), \( X_i \), entered in the PLYSHLOG table and calculates a corresponding viscosity multiplier, \( Z_i \), as follows:

\[
Z_i(X_i) = \frac{1 + (P - 1)M_i^{ref}(X_i)}{P}
\]

where

\( M_i^{ref} \) is calculated based on the reference conditions specified in PLYSHLOG.

\( P \) is the viscosity multiplier assuming no shear effect and is interpolated, based on the current polymer concentration (and salt and temperature, if applicable), from the relevant polymer viscosity keyword (PLYVISC, PLYVSCS, PLYVISCT or PLYVSCST). The value of \( P \) must be greater than or equal to unity.

Taking natural logarithms of the input velocities (or shear rates in the case of SHRATE), \( X_i \), and the corresponding viscosity multipliers, \( Z_i \), gives a discrete set of points (\( \ln X_i \), \( \ln Z_i \)) through which a piecewise linear function, \( f \), is constructed. Within this function, the straight line joining two consecutive points (\( \ln X_i \), \( \ln Z_i \)) and (\( \ln X_{i+1} \), \( \ln Z_{i+1} \)) may be written as:

\[
\ln X_i \ln Z_{i+1} + \ln \frac{X_{i+1}}{X_i} \ln Z = \ln Z_i \ln X_{i+1} + \ln \frac{Z_{i+1}}{Z_i} \ln X
\]

The shear water and shear polymer viscosities are then calculated as follows:

\[
\mu_{w,sh} = \mu_{w,eff}Z_w^*, \quad \mu_{p,sh} = \mu_{p,eff}Z_p^*
\]

where:

\( \mu_{w,sh} \) is the shear water viscosity following shear thinning/thickening

\( \mu_{p,sh} \) is the shear polymer viscosity following shear thinning/thickening

\( \mu_{w,eff} \) is the effective water viscosity

\( \mu_{p,eff} \) is the effective polymer viscosity
$Z_w^*$ is the shear effect multiplier that lies at the intersection of the piecewise linear function, $f$, and the straight line $\ln X_0 = \ln X + \ln Z$ where $X_0$ is the water phase flow velocity (or shear rate) prior to shear thinning/thickening.

$Z_p^*$ is as above albeit $X_0$ in this case is the polymer flow velocity (or shear rate) prior to shear thinning/thickening.

Shear thinning/thickening is not applied in those cases where:

- The value of $X_0$ is less than the minimum velocity (or shear rate) specified in PLYSHLOG.
- There is no significant effect due to the polymer concentration (case where $P = 1$)

In such cases, the shear viscosities are equal to the effective component viscosities, that is, $Z_w^* = Z_p^* = 1$.

Note that the data entered for PLYSHLOG must satisfy the constraint

$$\frac{d (\ln M_i^{\text{ref}})}{d \ln X} = \frac{\ln M_i^{\text{ref}} - \ln M_i^{\text{ref}}}{\ln X_{i+1} - \ln X_i} > -1$$

Notes

- If the shear effect is not significant, the PLYSHLOG keyword should be omitted, as the calculation of the velocity-dependent viscosity increases the run time.
- Models are sensitive to the shear data used and so care must be taken when using this keyword.
- In versions of ECLIPSE before 2012.1, shear thinning only was allowed. This has now been relaxed to allow shear thickening.

See the "Polymer flood model" in the ECLIPSE Technical Description for further information.

Example

With NTPVT=1 and NPPVT$\geq$6 specified in keyword TABDIMS and keyword PLYVSCST present:

```plaintext
PLYSHLOG
   -- reference conditions
   -- polymer          salinity          temperature
   0.5              0.0               0.0
   /                 
   -- velocity         multiplier
   -- m/day            dimensionless
   0.000001         1.0
   0.1              1.2
   1.0              1.6
   10.0             1.7
   100.0            1.8
   1000.0           1.8
   
```

Keywords
PLYTRRF  Polymer residual resistance factor temperature dependence

This keyword should only be used for polymer studies. It describes the temperature dependence of the residual resistance factor given by the second data item of keyword PLYROCK or by PLYKRRF.

The data comprises NTSFUN (see RUNSPEC keyword TABDIMS) tables of polymer residual resistance factor multiplier functions, each terminated by a slash (/), describing the effect on the residual resistance factor for adsorbed polymer due to increasing temperature. Each table consists of 2 columns of data:

1. Temperature.
   The values should increase monotonically down the column.
   UNITS: °C (METRIC), °F (FIELD), °C (LAB)

2. The corresponding factor by which the residual resistance factor (entered using the PLYROCK or PLYKRRF keywords) has to be multiplied to give the temperature-dependent residual resistance factor of adsorbed polymer.

If PLYTRRFA is specified, then for a given grid cell temperature T, the value of the temperature multiplier will be the value in column 2 for the first temperature in column 1 that is equal to or below T. Otherwise, the multiplier will be linearly interpolated as usual between the column 2 values for the column 1 temperatures that form the range containing T.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (RUNSPEC keyword TABDIMS). If PLYTRRFA is specified, then this number should also be one more than the value specified first argument of PLYTRRFA.

See "Polymer flood model" in the ECLIPSE Technical Description for further information.

Example

With NTSFUN=1 and NSSFUN≥5 specified in keyword TABDIMS:

```
PLYTRRF
  20.0  0.1
  50.0  0.5
 100.0  2.5
150.0  10.0
200.0  20.0 /
```
PLYTRRFA       Polymer residual resistance factor temperature
dependence controller

This keyword should only be used for polymer studies. It controls the way the temperature dependence of
the residual resistance factor given by PLYTRRF is interpreted.

If this keyword is absent, then PLYTRRF is interpreted as a piecewise linear specification of the
temperature-dependent multiplier of the residual resistance factor (RRF) of the adsorbed polymer. The RRF
is always calculated at the highest temperature experienced in the cell while polymer was present.

If this keyword is present, then PLYTRRF is interpreted as a histogram of this multiplier. The temperature
range is divided into intervals by PLYTRRF, and the adsorbed polymer in each cell is divided into batches
for each temperature interval.

This keyword has one data item:

1. NBTRRF: the number of intervals in the histogram version of PLYTRRF.

   NBTRRF must be greater than or equal to 2.

The keyword is terminated by a slash.

---

Note: There should be one more data line in PLYTRRF than in NBTRRF, to represent the upper limit of the
table. Hence NBTRRF =2 will require 3 data lines in PLYTRRF.

See keyword PLYTRRF and the description of the "Polymer flood model" in the ECLIPSE Technical
Description for further information.

Example

For a 2-interval representation:

```
PLYTRRFA
  2 /
```
**Polymer solution viscosity function**

The data comprises NTPVT (see RUNSPEC keyword TABDIMS) tables of polymer viscosity multiplier functions, each terminated by a slash (/), describing the effect on the viscosity of pure water of increasing the concentration of polymer in solution. Each table consists of 2 columns of data:

1. The polymer concentration in the solution.
   - The values should increase monotonically down the column.
   - The first entry should be 0.0
   - UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/sec (LAB), kg/sm$^3$ (PVT-M)

2. The corresponding factor by which the water viscosity (entered using the PVTW keyword in ECLIPSE 100 or the corresponding "Water components" keywords in ECLIPSE 300) has to be multiplied to give the viscosity of the solution.
   - The first entry should be 1.0.
   - The values should be level or monotonically increasing down the column.

This keyword should only be used for polymer studies where the salt-sensitivity option has not been activated (using keyword BRINE in RUNSPEC). However, if multi-component brine option is activated (using both BRINE and ECLMC keywords in RUNSPEC), then the keyword PLYVIS must be used as Multi-Component Brine model cannot currently be used with salt-sensitive Polymer model.

**Note:** If the surfactant option is active, the viscosity multiplier applies to the water viscosity supplied in the SURFVIS keyword.

The keyword may be used in the SCHEDULE section where the effect of polymer on viscosity may change during the simulation, for example when considering thermal degradation but should be used with care. The keyword must first have been defined in the PROPS section before it can be used in the SCHEDULE section. In addition, the keyword will affect all polymer in the reservoir, not just polymer injected after the keyword has been used.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (RUNSPEC keyword TABDIMS).

See "Polymer flood model" in the ECLIPSE Technical Description for further information.

**Example**

With NTPVT=1 and NPPVT≥5:

```
PLYVIS
.0000  1.0
.0003  10.0
.0005  25.0
.0008  50.0
.0010  80.0  /
```

**Keywords**
PLYVISCS  Polymer/salt solution viscosity function

This keyword should only be used for polymer studies when the salt-sensitivity option has been activated (in ECLIPSE 100 keyword BRINE in the RUNSPEC without ECLMC, in ECLIPSE 300 the use of this keyword will activate the salt-sensitivity option).

The data comprises NTPVT (see item 2 in RUNSPEC keyword TABDIMS) tables of polymer viscosity multiplier functions, each terminated by a slash (/), describing the effect on the viscosity of pure water of increasing the concentration of polymer and salt in the solution. Each table consists of up to NPPVT (see RUNSPEC keyword TABDIMS) sets of data records. Each set of records within a table consists of up to NPPVT records of data items to allow the specification of a full two-dimensional polymer/salt solution viscosity function. The first record of each elementary set of records contains the following items of data:

1. The polymer concentration in the solution.
   - The values should increase monotonically down the column.
   - The first entry should be 0.0
   - UNITS: kg/sm³ (METRIC), lb/stb (FIELD), gm/scc (LAB)

2. The corresponding factor by which the water viscosity (entered using the PVTW keyword in ECLIPSE 100 or the corresponding "Water components" keywords in ECLIPSE 300) has to be multiplied to give the viscosity of a fully mixed solution of polymer/salt at a polymer concentration given by item 1 and a salt concentration given by item 1 in the corresponding salinity table (SALTNODE keyword in ECLIPSE 100 or ESPNODE keyword in ECLIPSE 300).

   Note: The values should be greater than or equal to unity. Activating item 208 in OPTIONS relaxes this constraint by allowing any positive value.

For record 2 and all subsequent records up to a maximum of NPPVT, there should be only 1 data item in the record. This item defines the corresponding factor by which the pure water viscosity has to be multiplied to obtain the viscosity of the polymer/salt solution whose polymer concentration is given in item 1 above and whose salt concentration in specified in the corresponding record in the SALTNODE keyword in ECLIPSE 100 or in the ESPNODE keyword in ECLIPSE 300.

The data for each table is terminated by a null record (that is a record with no data before the terminating slash).

See "Polymer flood model" in the ECLIPSE Technical Description for further information.

Example

With NTPVT=1 and NPPVT=4:

```
PLYVISCS
.0000 1.0
   1.0
   1.0
   1.0 /
.0100 10.0
   8.0
   6.0
   4.0 /
.0200 25.0
   20.0
   15.0
   10.0 /
```

Keywords

ECLIPSE Reference Manual

1698
.0300 50.0
30.0
20.0
12.0 /
/
PLYVISCT  Polymer/temperature viscosity function

This keyword should only be used for polymer studies when the temperature option has been activated (keyword TEMP in the RUNSPEC section) and the salt-sensitivity option has not been activated (that is not using keyword BRINE in RUNSPEC).

The data comprises NTPVT (see item 2 in RUNSPEC keyword TABDIMS) tables of polymer viscosity multiplier functions, each terminated by a slash (/). The tables describe the effect of increasing temperature on the viscosity of pure water. Each table consists of up to NPPVT (see RUNSPEC keyword TABDIMS) sets of data records. Each set of records within a table consists of up to NPPVT records of data items to allow the specification of a full two-dimensional polymer viscosity function. The first record of each elementary set of records contains the following items of data:

1. The polymer concentration in the solution.
   The values should increase monotonically down the column.
   The first entry should be 0.0
   UNITS: kg/sm³ (METRIC), lb/stb (FIELD), gm/scc (LAB)

2. The corresponding factor by which the water viscosity (entered using the PVTW keyword) has to be multiplied to give the viscosity of a fully mixed solution of polymer at a polymer concentration given by item 1 and a temperature given by item 1 in the corresponding TEMPNODE table.
   The values should be greater than or equal to unity. Activating item 208 in OPTIONS relaxes this constraint by allowing any positive value.

For record 2 and all subsequent records up to a maximum of NPPVT, there should be only one data item in the record. This item defines the corresponding factor by which the pure water viscosity (entered using the PVTW keyword) has to be multiplied to obtain the viscosity of the polymer solution whose polymer concentration is given in item 1 above and whose temperature is specified in the corresponding record in the TEMPNODE keyword.

The data for each table is terminated by a null record (that is a record with no data before the terminating slash).

See the "Polymer flood model" in the ECLIPSE Technical Description for further information.

Example

With NTPVT=1 and NPPVT>4:

| PLYVISCT | 0.000  | 1.0  | 1.0  | 1.0  | /  |
|          | 0.010  | 10.0 | 8.0  | 6.0  | 4.0 | /  |
|          | 0.020  | 25.0 | 20.0 | 15.0 | 10.0 | /  |
|          | 0.030  | 50.0 | 30.0 | 20.0 | 12.0 | /  |
|          |        |     |     |     |     |    |

Keywords

PLYVISCT  Polymer/temperature viscosity function

ECLIPSE Reference Manual
**PLYVSCST**  

**Polymer/salt solution/temperature viscosity function**

This keyword should only be used for polymer studies when the salt-sensitivity option has been activated (keyword **BRINE** in the RUNSPEC section) and when the temperature option has been activated (keyword **TEMP** in the RUNSPEC section).

The data comprises NTPVT (see item 2 in RUNSPEC keyword **TABDIMS**) tables of polymer viscosity multiplier functions, each terminated by a slash (/). These describe the effect on the viscosity of pure water of increasing the concentration of polymer and salt in the solution, and the temperature of the solution.

Each table consists of up to NPPVT (see RUNSPEC keyword **TABDIMS**) sets of data records, one for each polymer concentration. The set of records then has NPPVT subsets, one for each brine concentration value specified by the **SALTNODE** keyword, with each subset containing NPPVT records for each temperature specified by the **TEMPNODE** keyword.

In outline, the record structure for one table is shown below:

<table>
<thead>
<tr>
<th>Column 1</th>
<th>Column 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polymer concentration value 1</td>
<td>multiplier at polymer concentration 1, brine concentration 1, and temperature 1</td>
</tr>
<tr>
<td>multiplier at polymer concentration 1, brine concentration 1, and temperature 2</td>
<td></td>
</tr>
<tr>
<td>multiplier at polymer concentration 1, brine concentration 1, and temperature 3 /</td>
<td>end of first record set (brine concentration 1) for polymer concentration 1</td>
</tr>
<tr>
<td>multiplier at polymer concentration 1, brine concentration 2, and temperature 1</td>
<td></td>
</tr>
<tr>
<td>multiplier at polymer concentration 1, brine concentration 2, and temperature 2</td>
<td></td>
</tr>
<tr>
<td>multiplier at polymer concentration 1, brine concentration 2, and temperature 3 /</td>
<td>end of 2nd record set (brine concentration 2) for polymer concentration 1</td>
</tr>
<tr>
<td>/</td>
<td>end of records for polymer concentration 1</td>
</tr>
<tr>
<td>Polymer concentration value 2</td>
<td>multiplier at polymer concentration 2, brine concentration 1, and temperature 1</td>
</tr>
</tbody>
</table>

Record sets repeat for remaining polymer concentration values

/ one additional slash to terminate this table

remaining tables (if any) have the same format

Table 3.28: PLYVSCST table structure for 2 SALTNODE values and 3 TEMPNODE values

In this table, there is only one column for the multiplier, except for the first record where the polymer concentration changes, where there are two columns: concentration and multiplier. *Italics show comments.*

Each elementary set of records refers to one polymer concentration and one salt concentration, and gives a single column of viscosity multipliers, one record for each temperature in the **TEMPNODE** keyword (except
where the polymer concentration changes, where the multiplier is in column 2). For successive sets of records, each set refers to a salt concentration specified by the SALTNODE keyword.

When the record set for the largest SALTNODE value has been completed, the next set of records refers to a new polymer concentration, and restarts at the first TEMPNODE and SALTNODE values. The multiplier is preceded by the polymer concentration value for this record only. In combination this allows the specification of a full three-dimensional polymer/salt solution/temperature viscosity function.

The multipliers should be greater than or equal to unity. Activating item 208 in OPTIONS relaxes this constraint by allowing any positive value.

Note that:

The polymer concentration in the solution should increase monotonically down the table
The first entry should be 0.0
UNITs: kg/sm³ (METRIC), lb/stb (FIELD), gm/scc (LAB)
The set of records for one polymer concentration is terminated by a null record (that is a record with no data before the terminating slash).

See the "Polymer flood model" in the ECLIPSE Technical Description for further information.

Example

With NTPVT=1 and NPPVT ≥ 3, for 2 SALTNODE and 3 TEMPNODE values (text after the slashes provides comments):

```
PLYVSCST
 .0000   1.0 
  1.0  
 1.0 / set of records for salt concentration value #1 
 1.0 
 1.0 / set of records for salt concentration value #2 
/ end of records for polymer concentration 0.0000 
 .0100  10.0 
  8.0  
 6.0 / set of records for salt concentration value #1 
 8.0 
 4.0 
 2.0 / set of records for salt concentration value #2 
/ end of records for polymer concentration 0.0100 
 .0300  50.0 
  25.0 
 15.0 / 
 40.0 
 30.0 
 18.0 / 
/ end of records for polymer concentration 0.0300 
/ end of table 1
```
Pressure maintenance region numbers

The keyword should be followed by one integer for every grid block in the current box specifying the pressure maintenance region to which it belongs. The region number should not be less than 1 or greater than the number of pressure maintenance tables, specified with TABDIMS.

Pressure maintenance region numbers are used to specify regions for pressure maintenance as described with the GPMAINT keyword.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices.

By default all cells are in PMANUM region 1.

Example

```
PMANUM
100*1 100*2 /
```
**PMAX**

**Maximum pressure in the simulation**

The keyword should be followed by some or all of the following items, indicating the maximum pressure likely to be reached during the simulation and additional information governing total compressibility checks. The record should be followed by a slash (/).

**Note:** In ECLIPSE 300, this keyword is active only for Blackoil and Thermal Blackoil runs with dissolved gas (RS) or vaporized oil (RV).

1. The maximum pressure likely to be reached during the simulation

**Note:** This item is ignored in ECLIPSE 300.

This value is used only if live oil PVT properties are entered using keyword PVCO. The information is used for two purposes. Firstly, the undersaturated parts of the oil PVT tables are automatically filled out in the form of the PVTO keyword, up to the pressure entered in PMAX. Secondly, ECLIPSE carries out a check to see if compressibilities are changing too rapidly with Pbub so that the curves of undersaturated oil formation volume factor for two different values of Rs cross each other. A warning is printed if they cross at a pressure below the maximum pressure entered in PMAX.

**UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

**DEFAULT:** ERROR: this item must be entered with PVCO tables.

2. Maximum pressure value to extend the range of pressures for checking total compressibility of the system.

By default the total compressibility of the gas-oil system is checked across the range of pressures spanned by the bubble point pressures in the PVTO (or PVCO) tables (or plain pressure nodes in a PVDO table) in dead oil cases; and by the range of dew point pressures in the PVTG tables (or plain PVDG tables) in dry gas cases. The upper pressure limit for the total compressibility checks is then set to the maximum of the value entered here and the maximum saturation pressure specified in the input tables. This could be useful to check total compressibility errors arising from extrapolation above the highest entered Rs value in the PVTO table which would cause convergence difficulties. It is not necessary to use PVCO to enable this checking, as this also applies to PVTO tables. See “Total compressibility checks” in the ECLIPSE Technical Description for more information.

**UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

**DEFAULT:** 0.0 barsa

3. Minimum pressure value to extend the lower range of pressures for checking total compressibility of the system.

By default the total compressibility of the gas-oil system is checked across the range of pressures spanned by the bubble point pressures in the PVTO (or PVCO) tables (or plain pressure nodes in a PVDO table) in dead oil cases; and by the range of dew point pressures in the PVTG tables (or plain PVDG tables) in dry gas cases. The lower pressure limit for the total compressibility checks is then set to the minimum of the value entered here and the minimum saturation pressure specified in the input tables.

**UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

**DEFAULT:** 1.0E+20 barsa
4. The number of nodes used to span the range of pressures for checking total compressibility. See "Total Compressibility Checks" in the ECLIPSE Technical Description for more information. Up to 100 nodes may be used.

DEFAULT: 30

**Example**

```
PMAX
7000 8000 /
```
### PMISC

#### Pressure-dependent miscibility tables

The `PMISC` keyword is optional, and may only be used in miscible runs (see keyword `MISCIBLE`). It supplies tables that control the transition between immiscible and miscible displacement as a function of oil pressure. If this keyword is not present in miscible runs, miscible displacement is assumed to occur at all pressures; pressure having no effect on the miscibility. See "Miscible flood modeling" in the ECLIPSE Technical Description. Saturation functions for the immiscible fluid must be specified when `PMISC` is used in a miscible run.

If used in runs which use the Solvent Model in miscible mode (keywords `SOLVENT` and `MISCIBLE` in the RUNSPEC section) then see also the section entitled "Using the Solvent Model" in the ECLIPSE Technical Description.

The data comprises `NTMISC` (see keyword `MISCIBLE`) tables of the pressure dependent miscibility function, each terminated by a slash.

Each table consists of 2 columns of data:

1. The local oil phase pressure.
   - Values should increase monotonically down the column.
   - UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
2. The corresponding miscibility, on a scale from 0.0 to 1.0.
   - Zero corresponds to immiscible displacement and 1.0 to miscible displacement.
   - The values should be level or increasing down the column.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than `NSMISC` (defined by keyword `MISCIBLE`).

#### Example

With `NTMISC=1` and `NSMISC≥4`:

```plaintext
PMISC
1000.0 0.0
4000.0 0.0
4100.0 1.0
8000.0 1.0 /
```

Keywords

1706
Poisson’s ratio for rock stress balance

This keyword has no effect if GEOMECH is not specified in the RUNSPEC section.

The keyword should be followed by one non-negative real number for every grid block in the current input box, specifying the rock Poisson ratio. This parameter is dimensionless and must be greater than -1 and less than 0.5. The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces may not be inserted on either side of the asterisk.

**Note:** There is no default for the Poisson ratio. The simulation will stop if the GEOMECH keyword is specified but this data is not supplied.

**Example**

```
POISSONR
324*0.3 /
```
POLYMER  Activates the Polymer Flood Model

This keyword activates the Polymer Flood Model (see "Polymer flood model" in the ECLIPSE Technical Description).

The keyword has no associated data.
PORO

Specifies the grid block porosity values

The keyword should be followed by one non-negative real number for every grid block in the current input box, specifying the fractional porosity value for each cell.

Grid blocks whose pore volume is zero are treated by the program as inactive. Since the computing time and storage space requirements of a run depend primarily on the number of active grid blocks, it is important that inactive blocks should be unambiguously identified. This is best done by setting either PORO or NTG to zero in inactive blocks. It is also possible to set pore volumes directly (array PORV in the EDIT section), or to use the ACTNUM keyword.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0.127). Note that spaces must not be inserted on either side of the asterisk.

The data must be terminated by a slash (/).

Note: For thermal runs both the rock volume and the pore volume should be zero for a cell to become inactive.

Examples

Example 1

|------- | IX1-IX2 | JY1-JY2 | KZ1-KZ2 |
| BOX | 5 | 16 | 3 | 8 | 2 | 3 | / |
| PORO | 0.16 | 0.14 | 0.12 | 0.1 | 0.08 | 0.09 | 0.1 | 0.11 | 0.12 | 0.13 | 0.14 | 0.14 |
| | 0.16 | 0.14 | 0.12 | 0.1 | 0.09 | 0.09 | 0.1 | 0.11 | 0.12 | 0.13 | 0.14 | 0.14 |
| | 0.15 | 0.14 | 0.12 | 0.1 | 0.10 | 0.09 | 0.1 | 0.11 | 0.12 | 0.13 | 0.14 | 0.14 |
| | 0.15 | 0.14 | 0.12 | 0.1 | 0.10 | 0.09 | 0.1 | 0.11 | 0.12 | 0.13 | 0.14 | 0.14 |
| | 0.14 | 0.14 | 0.12 | 0.1 | 0.09 | 0.09 | 0.1 | 0.11 | 0.12 | 0.13 | 0.14 | 0.14 |
| | 0.14 | 0.14 | 0.12 | 0.1 | 0.08 | 0.09 | 0.1 | 0.11 | 0.12 | 0.13 | 0.14 | 0.14 |
| | 0.15 | 0.15 | 0.12 | 0.1 | 0.09 | 0.09 | 0.1 | 0.11 | 0.12 | 0.13 | 0.15 | 0.15 |
| | 0.15 | 0.15 | 0.12 | 0.1 | 0.10 | 0.09 | 0.1 | 0.11 | 0.12 | 0.13 | 0.15 | 0.15 |
| | 0.15 | 0.15 | 0.12 | 0.1 | 0.10 | 0.09 | 0.1 | 0.11 | 0.12 | 0.13 | 0.15 | 0.15 |
| | 0.15 | 0.15 | 0.12 | 0.1 | 0.09 | 0.09 | 0.1 | 0.11 | 0.12 | 0.13 | 0.15 | 0.15 |
| | 0.15 | 0.15 | 0.12 | 0.1 | 0.08 | 0.09 | 0.1 | 0.11 | 0.12 | 0.13 | 0.15 | 0.15 |
| ENDBOX |

Example 2

In this example, porosity values for a 10*10 cross section are set, with an inactive third layer:

| PORO | 10*0.21 | 10*0.24 | 10*0.0 | 30*0.23 | 20*0.18 | 20*0.03 | / |

ECLIPSE 100

Every PORO value in the top plane (K = 1) must be specified in one way or another by the end of the GRID section. Values in lower planes (K > 1) which are not specified default to the value in the plane above.
**PORV**

**Specifies the grid block pore volumes**

The keyword should be followed by one non-negative real number for every grid block in the current input box, specifying the grid block pore volumes.

This keyword is entirely optional. Any pore volumes that are not altered remain at the values calculated by the program from the GRID data.

Grid blocks whose pore volume is zero are treated by the program as inactive. Since the computing time and storage space requirements of a run depend primarily on the number of active grid blocks, it is important that inactive blocks should be unambiguously identified. This can be done either by using the PORV keyword (described here) to overwrite pore volumes directly in the EDIT section, or by setting PORO or NTG to zero in the GRID section. The ACTNUM keyword (in the GRID section) can also be used to identify inactive grid blocks.

In ECLIPSE 100 only, grid cells defined as inactive in the GRID section could be activated in the EDIT section using the PORV keyword. A warning message is issued in this case.

In ECLIPSE 300 only, grid cells defined as inactive in the GRID section using the ACTNUM keyword cannot be activated in the EDIT section. The use of the PORV keyword is ignored in this case.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*195.8). Note that spaces must not be inserted on either side of the asterisk.

**UNITS:** rm$^3$ (METRIC), rb (FIELD), rcc (LAB), rm$^3$ (PVT-M)

The data must be terminated by a slash (/).

---

**ECLIPSE 100 only**

**Note:** If a cell is defined with zero pore volume in the GRID section, then making it active by changing the pore volume in the EDIT section is not allowed with the polymer, surfactant, foam, temperature or environmental tracer options.

---

**ECLIPSE 300 only**

**Note:** For thermal runs both the rock volume and the pore volume should be zero for a cell to become inactive.

**Note:** It is not possible to give a non-zero pore volume to a cell which has a thickness of zero. An error message will be issued in this case (this can be reduced to a warning in ECLIPSE 300 by setting item 233 in OPTIONS3 to a non-zero value). Individual cell thicknesses can be adjusted with ADDZCORN if needed.

---

**Example**

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
   11 14 3 8 6 6 /
PORV
   24*0 /
ENDBOX
```
PPCWMAX  Limits the calculated PCW values when SWATINIT is used

This keyword can be used to limit the scaling of the capillary pressure curves when the SWATINIT keyword has been used to input an initial water distribution. Typically, if the input water saturation is above the connate saturation in blocks well above the transition zone, the scaling of the $P_c$ curve leads to unphysically high capillary pressure values.

The keyword should be followed by NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) records, each terminated by a slash (/).

Each record consists of the following items of data:

1. The maximum capillary pressure for this saturation region; it must be greater than the maximum capillary pressure in the corresponding saturation function table.
   
   **UNITS:** bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   
   DEFAULT: Infinity (no limit)

2. Either YES or NO, indicating whether the connate saturation should be modified to honor the input water saturation when the capillary pressure limit is exceeded.

   DEFAULT: NO

If a maximum capillary pressure is supplied via item 1 and item 2 is set to NO, the input water saturation (from SWATINIT) is not honored in grid blocks that exceed the maximum capillary pressure. In such cases, the rescaled maximum capillary pressure will be set to the maximum specified by PPCWMAX, and the initial water saturation will be adjusted to be consistent with this rescaling.

If a maximum capillary pressure is supplied via item 1, and item 2 is set to YES, the capillary pressure is reset using the values input via the saturation tables, and the connate saturation will be set to the SWATINIT value. In such cases, the maximum table capillary pressure remains unscaled and the effect of changing the connate saturation to the supplied SWATINIT value will be to adjust the initial water saturation to be consistent with the unscaled capillary pressure.

See also the SWATINIT and PCW keywords in the PROPS section.

Example

With NTSFUN=2

<table>
<thead>
<tr>
<th>PPCWMAX</th>
<th>15.0</th>
<th>YES</th>
<th>/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>17.0</td>
<td>YES</td>
<td>/</td>
</tr>
</tbody>
</table>
PRCORR  

Request modified Peng-Robinson EoS

This keyword requests that a slightly modified form of the Peng-Robinson equation of state is to be used, as described in "Equations of state" in the ECLIPSE Technical Description. This changes the usual form of the $\Omega_a$ value as a function of the component acentric factor.

The keyword contains no items, and has no effect on equations of state other than the Peng-Robinson.
Reference pressures

This keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section).

In a live-oil thermal run with \( N_c \) components, \( \text{PREF} \) specifies the reference pressures at which the hydrocarbon liquid densities are defined.

The expression used to calculate oil component liquid densities is the following:

\[
\rho_c = \frac{\rho_{ref_c}}{\left[1 - c_{PEC} (P - P_{ref_c})\right]^2 + c_{T1C} (T - T_{ref_c})} \\
\text{Eq. 3.148}
\]

where

- \( \rho_{ref_c} \) is the reference density, specified with keyword DREF,
- \( c_{PEC} \) is the oil component liquid compressibility, specified with keyword CREF,
- \( P_{ref_c} \) is the reference pressure, specified with this keyword,
- \( T_{ref_c} \) is the reference temperature, specified with keyword TREF, and
- \( c_{T1C} \) is the thermal expansion coefficient, specified with keyword THERMEX1.

There should be one value for each component:

UNITs: barsa (METRIC), psia (FIELD), atms (LAB), atms (PVT-M)

DEFAULT: the standard pressure set with keyword STCOND.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

Example

```
\begin{verbatim}
PREF
1527.4
1527.4 /
\end{verbatim}
```
Reference pressures for surface EoS

This keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section).

This optional keyword supplies reference pressures for use in the surface calculations. The default, if it is not supplied, is for these to follow the reservoir values supplied with PREF.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

UNITs: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

Example

```
PREFS
1527.4
1527.4 /
```
**PREFT**

### Reference pressures

This keyword can only be used with the thermal option (keyword `THERMAL` in the `RUNSPEC` section).

`PREFT` is an alias for `PREF`. This keyword was introduced to facilitate export of thermal data from `PVT i`. It makes no difference to the simulator whether this keyword or the older `PREF` keyword is used.

<table>
<thead>
<tr>
<th>ECLIPSE 100</th>
<th></th>
<th>ECLIPSE 300</th>
<th></th>
<th>SPECIAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td></td>
<td>x</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**PREFTS**

Reference pressures for surface EoS

This keyword can only be used with the thermal option (keyword **THERMAL** in the **RUNSPEC** section). **PREFTS** is an alias for **PREFS**. This keyword was introduced to facilitate export of thermal data from PVT $i$. It makes no difference to the simulator whether this keyword or the older **PREFS** keyword is used.
**PREFW**

Reference pressures for water components

**PREFW** specifies the reference pressure used in calculating the water density and water viscosity for each of the \( N_w \) water components.

The water component densities, at pressure \( P \), are calculated from:

\[
\rho_w = \rho_{ref_w} \cdot \left( 1 + X + X^2 / 2 \right)
\]

\[
X = c_{P_w} \cdot \left( P - P_{ref_w} \right)
\]

where

- \( \rho_{ref_w} \) is the reference density, specified with keyword **DREFW**,
- \( c_{P_w} \) is the water component compressibility, specified with keyword **CREFW**
- \( P_{ref_w} \) is the reference pressure, specified with this keyword

The water component viscosities are calculated from:

\[
\mu_w = \mu_{ref_w} \cdot \left( 1 + Y + Y^2 / 2 \right)
\]

\[
Y = -c_{V_w} \cdot \left( P - P_{ref_w} \right)
\]

where

- \( \mu_{ref_w} \) is the reference viscosity, specified with keyword **VREFW**
- \( c_{V_w} \) is the viscosibility, specified with keyword **VREFW**

There should be one value for each component:

**UNITS:** barsa (**METRIC**), psia (**FIELD**), atma (**LAB**), atma (**PVT-M**)

**DEFAULT:** 1 atma.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword **TABDIMS**. The use of multiple EoS regions is discussed in "Equations of state regions" in the **ECLIPSE Technical Description**.

See "Water components" in the **ECLIPSE Technical Description** for further details of the water component option.

**Example**

```
PREFW
  1.0 2.0 /  
```

---

**Keywords**

1717
Reference pressures of water components for surface EoS

This optional keyword supplies reference pressures for water components for use in the surface calculations. The default, if it is not supplied, is for these to follow the reservoir values supplied with PREFW.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

UNITS: barsa (METRIC), psia (FIELD), atm (LAB), atma (PVT-M)
Initial pressures

The keyword should be followed by one real number for every grid block specifying the initial pressure. The data field should be terminated by a slash (/).

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*7200). Note that spaces must not be inserted on either side of the asterisk (*).

This keyword must be used with other keywords (such as SWAT, SGAS, XMF and YMf) sufficient to define the initial state explicitly for the run

Example

With NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS:

```
PRESSURE
  240*4927
```
This keyword is used to switch on the well prioritization option, and to set coefficients in the priority equations.

This method of limiting the group production rate is an alternative to the method of apportioning group production targets among the wells in proportion to their guide rates. In the prioritization option, wells are turned on in decreasing order of priority, the well with the highest priority flowing first, until a group’s production rate limit is exceeded. The flowing wells operate at their individual targets or limits (see the keyword \texttt{WCONPROD}). But the well that exceeds the group’s rate limit is cut back to meet the limit, and the wells with priority lower than this subordinate to that group do not flow. The low priority wells not selected for flow are priority-closed until they are eventually selected. In the well reports, priority-closed wells are indicated by “PSHT” or “PSTP” depending on whether they are shut or stopped, as requested in item 9 of keyword \texttt{WELSPECS}. Wells that are closed manually by the user, or by violating economic limits, cannot be selected for flow, whatever their priority number is.

Optionally, the coefficients for a second priority formula can also be defined. In the \texttt{GCONPRI} keyword it is possible to select which of the two priority formulae should be used to obey each rate limit separately. For example, if a gas rate limit is exceeded you may wish to priority-close wells with a high GOR, whereas to obey a water rate limit the user may wish to close wells with a high water-cut. When there are two priority formulae defined, the procedure is slightly different to the single priority equation case. All the available (open or priority-closed) wells are opened initially, then if a group rate limit is violated ECLIPSE closes in turn the wells with the lowest priority according the formula chosen for that particular rate limit until the rate limit is no longer exceeded. The last well to be closed is then reopened and made to produce at a rate calculated to meet the limit. If the group exceeds two or more limits having different priority formulae, ECLIPSE determines which limit is broken by the largest fraction and uses the formula appropriate to that limit to select the lowest priority well to close. As each well is closed, the active formula may, of course, change.

The wells are prioritized at each Newton iteration of each timestep, in the first \texttt{NUPCOL} iterations. Any wells that flow with a reduced target calculated to meet a group’s rate limit have their target rates recalculated to meet the group’s limit exactly. During any subsequent iterations of each timestep, the on-off status and flow target of each well will be kept unchanged, to allow the iteration to converge. The value of \texttt{NUPCOL} is set by keyword \texttt{NUPCOL}.

While a well has its flow rate cut back to meet a group’s limit, its minimum rate limits set in keywords \texttt{WECON} and \texttt{WLIFT} are ignored. This is to prevent the well being prematurely closed or put on artificial lift when it is producing at a low rate so as not to exceed the group’s production rate limit.

The priority numbers for each well are calculated from their potential production rates (see “Well potentials” in the \textit{ECLIPSE Technical Description}) at regular intervals, using the general formula

\[
\text{Priority} = \frac{A + B Q_o + C Q_w + D Q_g}{E + F Q_o + G Q_w + H Q_g}
\]

\text{Eq. 3.149}

where

\(Q_o, Q_w\) and \(Q_g\) are the well’s potential oil, water and gas production rates, and

\(A - H\) are 8 coefficients set with this keyword.

The equation allows the priority to be set equal to, for example, the potential oil rate, or the reciprocal of the potential gas rate, or the reciprocal of the water cut. Individual wells may also have their priorities set independently by the user, overriding the general formula (see keyword \texttt{WELPRI}).
The prioritization option can be applied to group and field production control, but not injection control (which is still handled by guide rates). When this option is used, the group production rate limits should be set using the keyword \texttt{GCONPRI}, not \texttt{GCONPROD}. Group limits can be applied at any level in the grouping tree.

It is possible to mix the two group control strategies - prioritization and guide rates - in a single run. Groups to be controlled by prioritization should have their rate limits set with keyword \texttt{GCONPRI}, while groups using the guide rate method have their rate limits set with keyword \texttt{GCONPROD}. A group can be changed from one method to the other simply by re-specifying its rate limits with the appropriate keyword. The mixing of the two methods, however, is subject to the following restriction: the guide rate method cannot be used by a group subordinate to a group using the prioritization method. Thus, for example, no groups can use the guide rate method if the field has a prioritization rate limit.

When the two group control strategies are mixed, the well rates are allocated according to the following method. First, the production wells subordinate to prioritization groups are solved, selecting the wells for flowing in decreasing order of their priorities. Next, the producers in the remaining parts of the grouping tree are solved, employing the standard guide rate method to apportion the flow rates between them to meet any \texttt{GCONPROD} group targets. If a prioritization group is subordinate to one with a \texttt{GCONPROD} rate target, the wells in the prioritization group already have had their flow rates set, and the remaining wells have their rates set to make up the rest of the \texttt{GCONPROD} group’s target. Wells from the drilling queue (keyword \texttt{QDRILL} or \texttt{WDRILPRI}) that are not subordinate to a prioritization group will be opened when necessary to meet the \texttt{GCONPROD} group’s rate target. The drilling queue is not used for opening wells subordinate to prioritization groups, as the prioritization algorithm itself handles the opening and closing of these wells.

The \texttt{PRIORITY} keyword is followed by some or all of the following items of data, terminated with a slash (/). The record may be terminated early with a slash; the remaining items assume their default values. The items are:

- **Item 1**: The minimum time interval between well priority calculations
  
  Wells have their priorities calculated from the formula above at the beginning of each timestep that starts after the specified interval has elapsed since the previous calculation.
  
  A zero value causes the priorities to be recalculated every timestep. This may result in oscillations with some wells alternately switching on and off.
  
  **UNITS**: day (\texttt{METRIC}), day (\texttt{FIELD}), hr (\texttt{LAB}), day (\texttt{PVT-M})

- **Items 2 - 9**: The coefficients A - H in the first priority formula.
  
  None of the coefficients can have negative values.
  
  At least one of the first four coefficients (A - D) must be non-zero, and similarly at least one of the last four coefficients (E - H) must be non-zero.
  
  **DEFAULT**: Zero

  \textit{ECLIPSE 100 only}

- **Items 10 - 17**: The coefficients A - H in the second priority formula.
  
  None of the coefficients can have negative values.
  
  A second priority formula will be defined if one or more of these coefficients is non-zero. In this case, at least one of the first four coefficients (A - D) must be non-zero and at least one of the last four coefficients (E - H) must be non-zero.
  
  **DEFAULT**: Zero
Examples

Example 1
Recalculate priority numbers after every 100 days. Open wells in decreasing order of oil potential.

```
PRIORITY
100  0.0  1.0  0.0  0.0  1.0  0.0  0.0  0.0  /
```

Example 2
Recalculate priority numbers at each timestep. Open wells from lowest to highest water-liquid ratio.

```
PRIORITY
0  0.0  1.0  1.0  0.0  0.0  0.0  1.0  0.0  /
```

Example 3
Recalculate priority numbers at each timestep. Use two priority formulae, the first favoring wells with low water-cut and the second favoring wells with low GOR.

```
PRIORITY
0  0.0  1.0  1.0  0.0  0.0  0.0  1.0  0.0
0.0  1.0  0.0  0.0  0.0  0.0  0.0  1.0  /
```
PROPFLOW  Proppant flowback in a fracture

This keyword is used to provide a prediction of proppant flowback in a fracture. Two proppant flowback stability criteria are available.

This keyword is only available to Schlumberger engineers and can only be accessed if the Proppant Flowback license is obtained.
PRORDER  Order of applying group production rules

This keyword can optionally be used to define a specific sequence of actions to be performed if a group fails to meet its production rate target set in keyword **GCONPROD**. A full description of these actions can be found under "Group production rules" in the **ECLIPSE Technical Description**.

The keyword consists of two records of data, each terminated by a slash (/).

**Record 1**

This should contain a list of some or all of the following action mnemonics, terminated with a slash. The order in which the mnemonics are entered determines the order in which the actions are performed. Thus, whenever a group cannot make its **GCONPROD** production rate target, initially the first action in the list is performed. When no more actions of this type can be performed, **ECLIPSE** commences the second action in the list, and so on. The actions are:

- **DRILL**  Open new wells from the drilling queue.
  The drilling queue must be set up with either keyword **QDRILL** or **WDRILPRI**, containing production wells declared as **SHUT** in keyword **WCONPROD**. Restrictions on the rate at which wells can be drilled may be imposed by declaring the time taken to drill each well (keyword **WDRILTIM**) and defining drilling rigs (keyword **GRUPRIG**). Wells in the Sequential Drilling Queue (keyword **QDRILL**) will be drilled in the sequence in which they are placed in the queue. Wells in the Prioritized Drilling Queue (keyword **WDRILPRI**) are drilled in decreasing order of their drilling priority. The order in which wells are drilled may be affected by drilling rig constraints.

- **REPERF**  Open new connections on **AUTO** in existing wells.
  The connections must be placed on automatic opening by specifying 'AUTO' in item 6 of keyword **COMPDAT** (or in keyword **WELOPEN**). Wells are worked over in turn to open these connections. Restrictions on the rate at which wells can be worked over may be imposed by declaring the time taken to perform a workover (keyword **WORKLIM**) and defining workover rigs (keyword **GRUPRIG**). When a group needs more production, **ECLIPSE** will examine its subordinate wells that have connections on **AUTO** and are not prevented from being worked over by rig constraints. The well with the largest sum of [connection factor * phase mobility] of its remaining connections on **AUTO** is selected, and the next connection in its **AUTO** queue is opened.

- **THP**  Reduce each well's THP limit to a specified lower value.
  Secondary well THP limits should be specified in item 8 of keyword **WLIFT**. If a decrement value is entered in item 11 of **WLIFT**, the well’s THP is decreased to its secondary limit in stages, a decrement at a time. When a group needs more production, **ECLIPSE** examines its subordinate wells operating on THP control that have secondary THP limits waiting to be applied. The well that gives the biggest increase in flow when switched to its secondary THP limit (or when given its next THP decrement towards this limit) is selected, and its current THP limit is replaced by its secondary THP limit (or decremented by the amount specified in item 11).

- **RETUBE**  Change VFP table numbers (for example to model a retubing operation).
  Secondary VFP table numbers should be specified for wells in item 4 of keyword **WLIFT**. When a group needs more production, **ECLIPSE** will examine its subordinate wells operating
on THP control that have secondary VFP tables waiting to be used. The well that gives the biggest increase in flow when switched to its secondary VFP table is selected, and its current VFP table number is replaced by its secondary VFP table number.

**LIFT**

Change ALQ values (that is apply a fixed amount of artificial lift).

Secondary ALQ values should be specified for wells in item 5 of keyword WLIFT. If an increment value is entered in item 10 of WLIFT, the well’s ALQ is increased to its secondary value in stages, an increment at a time. A new well efficiency factor can also be defined in item 6 of keyword WLIFT if required, which comes into effect when the new ALQ value (or the first ALQ increment) is applied. Restrictions on the total amount of lift applied in any group may be specified with keyword GLIFTLIM. When a group needs more production, ECLIPSE examines its subordinate wells operating on THP control that have secondary ALQ values waiting to be applied and are not prevented from applying them by group lift capacity constraints. The well that gives the biggest increase in flow when switched to its secondary ALQ value (or when given its next ALQ increment towards this value) is selected, and its current ALQ value is replaced by the new value (or incremented by the amount specified in item 10).

The effect of this mnemonic is different if the Gas Lift Optimization facility is being used (see “Gas Lift Optimization” in the ECLIPSE Technical Description). Secondary ALQ values must not be specified with WLIFT, and GLIFTLIM must not be used. Instead, ECLIPSE automatically allocates lift gas to the wells until group production targets are met or other operating limits are reached. Gas lift allocations are calculated at specified intervals, or during the first NUPCOL Newton iterations of each timestep, depending on the instructions in keyword LIFTOPT, (regardless of the presence or absence of the LIFT mnemonic in PRORDER). By default, all the requested production rule actions will be attempted before gas lift is allocated. However, if LIFT is entered in the PRORDER keyword, gas lift allocation takes precedence over any production rule actions whose mnemonics are entered after LIFT. Thus, for example, it is possible to specify that drilling new wells should only be attempted if group targets cannot be met by applying gas lift. Essentially, all production rule actions entered after LIFT are only performed immediately after a gas lift optimization calculation, and not at any other time or Newton iteration, if a group production target still cannot be met. In this case it is best to configure LIFTOPT so that gas lift is optimized during each of the first NUPCOL iterations of every timestep.

**COMP**

Switch on, or increase power of, compressors linked to this group.

This option is only available if the Extended Network Model is in use (see "The extended network model" in the ECLIPSE Technical Description). The compressors must be defined with the NETCOMPA keyword, to be turned on when a nominated group cannot meet its target production rate of a specified phase. When the group needs more production, ECLIPSE turns on the compressors responding to the specified phase production of that group. If the group has more than one compressor responding to it, the compressors are turned on in increasing order of their switching sequence number (item 12 of keyword NETCOMPA). Multi-level compressors (see NETCOMPA item 10) have their power increased by one stage at a time until the group’s rate target is satisfied or the compressor is fully on and another compressor is tried.

A blank record, containing just the terminating slash, results in none of the above remedial actions being performed when a group cannot make its target rate (except for gas lift optimization, if this facility has been turned on). No drilling is performed, even if a drilling queue has been set up.
Record 2

The second record consists of these data flags, terminated by a slash (/). These are flags that control the application of the production rules. Both may be defaulted.

1. This item is a flag to control the application of the production rules when wells are treated as being shut for a given length of time during drilling or workovers (that is when item 3 of keyword WDRILTIM is set to YES). This flag has no effect if WDRILTIM item 3 is set to NO for all producers.

   The item consists of the characters YES or NO, specifying whether or not further actions are allowed before the full benefits of a DRILL or REPERF action are felt.

   YES  Allow further actions to take place in the timestep while a well is shut during a drilling or workover process, if the group cannot immediately make its target rate. This may result in further actions being performed before they are permanently needed.

   NO   When a production well is drilled or worked over to increase the flow rate in a group, and the well is treated as shut for the (non-zero) duration of this process, do not allow any further production rule actions to take place within this group until the next timestep that begins after the process has been completed.

   Setting this item to NO has a similar effect to the QDRILL command in keyword WDRILTIM, except that this item restricts all the group production rule actions and not just drilling.

   DEFAULT: YES

2. This item is a flag to control the order in which the actions in keyword WLIFT are performed when triggered by individual wells rather than by group behavior (that is when a well dies under THP control it will be revived after an action is performed, or an action can be performed when a well’s flow rate or water cut violates a limit specified in item 2 or 7 of keyword WLIFT). The actions may either be performed in the same sequence as specified in record 1 of keyword PRORDER, or in the original sequence given in WLIFT.

   The item consists of the characters YES or NO, specifying whether the original order in WLIFT should be overridden.

   YES  Apply the three actions (lower the THP, retube or apply lift) in the same order as specified in record 1 of keyword PRORDER.

   NO   Apply the actions in the original order given in keyword WLIFT, that is first THP, then RETUBE, then LIFT.

   DEFAULT: YES

Notes

- The production rules apply to all groups (including the FIELD) that have guide rate group control targets (keyword GCONPROD). They do not apply to groups under prioritization control (keyword GCONPRI).

- With the THP, RETUBE and LIFT actions, secondary values of the THP, VFP table number and ALQ must be supplied in keyword WLIFT. The WLIFT keyword will also apply these values to wells individually when their production rate falls below a specified limit, or their water cut or gas-liquid ratio exceeds a specified limit, or when the well dies under THP control. If you wish these secondary
values to be applied only via the group production rules to help meet a group target, or if a well dies under THP control, then you must give the rate, water cut and GLR limits in items 2, 7 and 9 of keyword \texttt{WLIFT} a default, zero or negative value.

- The \texttt{THP} action has no effect within groups that are part of a network (see "Network option" in the \textit{ECLIPSE Technical Description}), since the wells have their THP limits set according to their group's nodal pressure.

- If the gas lift optimization facility is being used (see "Gas Lift Optimization" in the \textit{ECLIPSE Technical Description}), by default all the requested production rule actions are attempted before gas lift is allocated. But if \texttt{LIFT} is entered in Record 1, any production rule actions entered after \texttt{LIFT} will only be performed immediately after a gas lift optimization calculation, if a group production target still cannot be met. The frequency of gas lift optimization calculations is governed by the \texttt{LIFTOPT} keyword, and is not affected by the presence or absence of the \texttt{LIFT} mnemonic in \texttt{PRORDER}.

- If the \texttt{PRORDER} keyword is not present then, for back compatibility, only the \texttt{DRILL} action will be applied (if a drilling queue has been set up), followed by the \texttt{COMP} action (in an Extended Network Model containing automatic compressors).

- If compressors have been defined in the extended network model with the \texttt{NETCOMPA} keyword (see "The extended network model" in the \textit{ECLIPSE Technical Description}), they are used regardless of whether the \texttt{PRORDER} keyword is entered or if the \texttt{COMP} option is missing from the \texttt{PRORDER} keyword. By default, the compressors are used as a last resort to increase a group's production rate if it fails to meet its target. Once a compressor has been defined, then, it is ultimately used when more flow is required; the \texttt{PRORDER} keyword just instructs ECLIPSE to turn the compressors on before some of the other options to increase the flow have been tried.

- When the network option is in use, by default all actions except for \texttt{COMP} will be performed in any network balancing iteration when a flow target is not met. Thus it is possible, for example, that a well may be opened from the drilling queue prematurely because an intermediate network balancing iteration has high nodal pressures. The exception is the \texttt{COMP} action, which is only performed if more flow is required once the network has converged. It is possible to instruct ECLIPSE to prevent any of the other production rule actions from being performed before the network balancing iters have converged, by using the \texttt{WAITBAL} keyword (see "Network option" in the \textit{ECLIPSE Technical Description}). Then if more flow is required once the network has converged, an appropriate production rule action is performed and the network must be balanced again. Using this option therefore requires more network balancing iterations in total.

### Example

```plaintext
PRORDER
  DRILL THP REPERF / 
  NO / 
```

causes the following actions to take place whenever a group cannot make its \texttt{GCONPROD} production target. Firstly, ECLIPSE searches the drilling queue for a suitable well to open. If no wells can be drilled, it examines the wells having secondary THP values and reduce the THP of the well that it selects. If it cannot reduce any THP values, it examines the wells having connections on AUTO opening, and opens the next AUTO connection in the well that it selects.
Initial pressure versus depth tables

This keyword can be used as an alternative to the PRESSURE keyword to specify the initial pressure for an enumerated initial solution.

The data comprises NTEQUL (see keyword EQLDIMS) tables of initial oil pressure versus depth, one for each equilibration region. Each table consists of 2 columns of data, and must be terminated by a slash (/).

1. Depth values.
   
   The values should increase monotonically down the column.

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

2. The corresponding values of oil pressure.

   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NDRXVD (see keyword EQLDIMS).

See also keywords SWAT, SGAS, RS, PBUB, RV and PDEW in the SOLUTION section.

Example

With NTEQUL=1 and NDRXVD≥4:

```
PRVD
7000  2310.0
8000  2640.0
9000  2990.0
10000 3390.0 /
```
Activate the generalized pseudo-pressure option in ECLIPSE 300

This keyword requests that the Generalized Pseudo-Pressure (GPP) option be activated for the well inflow calculations in ECLIPSE 300.

The keyword should be specified early in the \texttt{SCHEDULE} section before any well operations keywords or timestepping keywords are specified. Alternatively this option can be activated for individual wells by entering \texttt{GPP} in item 8 of \texttt{WELSPECS}.

For details of the GPP model, see "Special inflow equations" in the \textit{ECLIPSE Technical Description}.

See also keyword \texttt{PICOND}.
Primary residual gas saturation in gas-oil system

This keyword specifies the residual gas saturation in a gas-oil system within each grid cell for the primary saturation tables. This is only required when the ODD3P option is being used and when saturation table end-point scaling has been specified by the ENDScale keyword in the RUNSPEC section.

Each PSGRO keyword should be followed by one real number for each grid cell in the current input box specifying the residual saturation within the grid cell. The saturation should be in the range of 0.0 to 1.0 inclusive. If the value of PSGRO is omitted for any grid cell in a model for which ENDScale has been specified, the residual saturation will be obtained from the critical saturation in the appropriate saturation table.

See "Saturation table scaling" in the ECLIPSE Technical Description for further details.

See also the PSORG, PSORW, PSWRO, PSGRW and PSWRG keywords.

Example

With NDIVIX=3, NDIVIY=4 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS and no input BOX set:

```
PSGRO
60*0.1/
```
**PSGRW**  

**Primary residual gas saturation in gas-water system**

This keyword specifies the residual gas saturation in a gas-water system within each grid cell for the primary saturation tables. This is only required when the **ODD3P** option is being used and when saturation table end-point scaling has been specified by the **ENDSCALE** keyword in the **RUNSPEC** section.

Each **PSGRW** keyword should be followed by one real number for each grid cell in the current input box specifying the residual saturation within the grid cell. The saturation should be in the range of 0.0 to 1.0 inclusive. If the value of **PSGRW** is omitted for any grid cell in a model for which **ENDSCALE** has been specified, the residual saturation will be obtained from the critical saturation in the appropriate saturation table.

See "Saturation table scaling" in the *ECLIPSE Technical Description* for further details.

See also the **PSORG**, **PSGRO**, **PSORW**, **PSWRO** and **PSWRG** keywords.

**Example**

With **NDIVIX=3, NDIVIY=4** and **NDIVIZ=5** specified in **RUNSPEC** keyword **DIMENS** and no input BOX set:

```
PSGRW
60*0.1/
```
PSORG

Primary residual oil saturation in oil-gas system

This keyword specifies the residual oil saturation in an oil-gas system within each grid cell for the primary saturation tables. This is only required when the ODD3P option is being used and when saturation table end-point scaling has been specified by the ENDScale keyword in the RUNSPEC section.

Each PSORG keyword should be followed by one real number for each grid cell in the current input box specifying the residual saturation within the grid cell. The saturation should be in the range of 0.0 to 1.0 inclusive. If the value of PSORG is omitted for any grid cell in a model for which ENDScale has been specified, the residual saturation will be obtained from the critical saturation in the appropriate saturation table.

See "Saturation table scaling" in the ECLIPSE Technical Description for further details.

See also the PSGRO, PSORW, PSWRO, PSGRW and PSWRG keywords.

Example

With NDIVIX=3, NDIVIY=4 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS and no input BOX set:

```
PSORG
  60*0.1/
```
PSORW

Primary residual oil saturation in oil-water system

This keyword specifies the residual oil saturation in an oil-water system within each grid cell for the primary saturation tables. This is only required when the ODD3P option is being used and when saturation table end-point scaling has been specified by the ENDScale keyword in the RUNSPEC section.

Each PSORW keyword should be followed by one real number for each grid cell in the current input box specifying the residual saturation within the grid cell. The saturation should be in the range of 0.0 to 1.0 inclusive. If the value of PSORW is omitted for any grid cell in a model for which ENDScale has been specified, the residual saturation will be obtained from the critical saturation in the appropriate saturation table.

See "Saturation table scaling" in the ECLIPSE Technical Description for further details.

See also the PSORG, PSGRO, PSWRO, PSGRW and PSWRG keywords.

Example

With NDIVIX=3, NDIVY=4 and NDIVZ=5 specified in RUNSPEC keyword DIMENS and no input BOX set:

```
PSORW
60*0.1/
```
PSPLITX  

Specifies divisions of the x-direction for parallel runs

This is an optional keyword associated with the PARALLEL keyword. It specifies possible partitions arising from domain decomposition in the X-direction. It is envisaged that the data will be collected from the output of the PART argument in the RPTSOL keyword, which outputs all potential splits for a given dataset.

This keyword is a multiple record version of the now-redundant PDIVX keyword. The purpose of this keyword is to specify all possible partitions in a particular direction without having to edit the dataset.

The keyword is followed by multiple records each having N_{px} integer items (where N_{px} is the number of processors in the X-direction; if the number of processors specified by the NPROCX keyword matches the entries in a particular record, then this record is used for domain decomposition). Each integer specifies the last x-index for the particular processor. The last item must have the value N_{x}.

See also "Parallel option" in the ECLIPSE Technical Description.

Example

```plaintext
DIMENS 30 20 3 /
PSPLITX
15 30   / --NPROCX= 2 LOAD BALANCED (%) = 95.41
11 19 30 / --NPROCX= 3 LOAD BALANCED (%) = 94.59
 9 15 21 30 / --NPROCX= 4 LOAD BALANCED (%) = 92.58
 8 13 18 23 30 / --NPROCX= 5 LOAD BALANCED (%) = 91.04
 7 11 15 19 24 30 / --NPROCX= 6 LOAD BALANCED (%) = 91.04
 7 11 14 17 21 25 30 / --NPROCX= 7 LOAD BALANCED (%) = 82.14
 6 10 13 16 19 22 26 30 / --NPROCX= 8 LOAD BALANCED (%) = 76.94
 6 9 12 15 18 21 24 27 30 / --NPROCX= 9 LOAD BALANCED (%) = 80.93
 5 8 11 14 16 19 22 25 27 30/ --NPROCX= 10 LOAD BALANCED (%) = 72.83
/
```

Keywords
PSPLITY

Specifies divisions of the y-direction for parallel runs

This is an optional keyword associated with the PARALLEL keyword. It specifies possible partitions arising from domain decomposition in the Y-direction. It is envisaged that the data will be collected from the output of the PART mnemonic in the RPTSOL keyword, which outputs all potential splits for a given dataset.

This keyword is a multiple record version of the now redundant PDIVY keyword. The purpose of this keyword is to allow the user to specify all possible partitions in a particular direction without having to edit the dataset.

The keyword is followed by multiple records each having Npy integer items (where Npy is the number of processors is the Y-direction; if the number of processors specified by the NPROCY keyword matches the entries in a particular record, then this record will be used for domain decomposition). Each integer specifies the last y-index for the particular processor. The last item must have the value Npy.

See also "Parallel option" in the ECLIPSE Technical Description.

Example

```
DIMENS 30 20 3 /
PSPLITY
11 20 / --NPROCY= 2 LOAD BALANCED (%) = 94.59
  9 14 20 / --NPROCY= 3 LOAD BALANCED (%) = 84.69
  7 11 15 20 / --NPROCY= 4 LOAD BALANCED (%) = 94.18
  6 10 13 16 20 / --NPROCY= 5 LOAD BALANCED (%) = 79.45
  6 9 12 15 17 20 / --NPROCY= 6 LOAD BALANCED (%) = 80.04
  5 8 11 13 15 17 20 / --NPROCY= 7 LOAD BALANCED (%) = 70.94
  5 8 10 12 14 16 18 20 / --NPROCY= 8 LOAD BALANCED (%) = 72.83
/
```
Saturation pressure state assignment

This keyword uses the saturation pressure calculation to assign single phase hydrocarbon states as oil or gas, rather than the Li correlation. This gives a more precise state identification although this involves the extra cost of the $P_{\text{sat}}$ evaluation.

There is no data associated with this keyword.
PSTEADY  Turns on the pseudo steady-state calculation

This keyword advances the simulator to a steady state. Read the notes at the end of the keyword. The keyword is followed by the following items of data, denoting the day, month and year of the date required for the end of the simulation. This is the date that is given to the restart file, from which further predictive runs take place; and it is only used for output to the restart file. Further data specifies the criteria for the determination of whether steady-state has been reached. The keyword is terminated with a slash (/).

1. Day
   Day of the month (an integer between 1 and 31).

2. Month
   Name of the month (JAN, FEB, MAR, APR, MAY, JUN, JUL, AUG, SEP, OCT, NOV, DEC).
   For July, the abbreviation JLY is a valid alternative.

3. Year
   The year (a four-figure integer).
   DEFAULT: Is the date given in the START keyword if that has been given previously? There is no default if the START keyword is not present before this keyword; this item cannot be defaulted in this situation.

4. Measure of ‘closeness’ to steady-state where i refers to grid blocks, PV is the pore volume and Q represents all of pressure and the three saturations (if appropriate) and the scaling factor S is 1 for the saturations and brine and is the field average pressure for the pressure.

\[
\text{MAXABS} \quad \frac{\max_i \left| Q_i^{n+1} - Q_i^n \right|}{S \cdot \Delta t}
\]

\[
\text{L2} \quad \sqrt{\sum_i \left( PV(Q_i^{n+1} - Q_i^n) \right)^2} \quad \Delta t \sum_i PVQ_i^{n+1}
\]

DEFAULT: MAXABS

5. Tolerance for determining if pressure has reached steady-state.
6. Tolerance for determining if the oil saturation has reached steady-state.
7. Tolerance for determining if the water saturation has reached steady-state.
8. Tolerance for determining if the gas saturation has reached steady-state.
9. Tolerance for determining if the brine concentration has reached steady-state.
   DEFAULT: If the chosen norm is MAXABS the defaults are 1.0 e^{-3} for the saturations and brine concentration and 1.0 e^{-2} for pressure. If the norm is L2 then the defaults are all 1.0 e^{-6}.
10. Upper limit for timestepping to ensure that the calculation doesn’t run for ever if the steady state not found. The run will end when the simulation reaches this time.
   DEFAULT: 365000

UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
11. Lower limit for timestepping to ensure that the calculation doesn’t end too soon. The run will not end before the simulation reaches this time.

   DEFAULT: 0

   UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

12. Option to improve the initial equilibration of the model when analytic aquifers, such as AUCHWAT, AUCHGAS, AUFETP and AQUT, are used in the solution section. This option should help to obtain a better estimate of the steady-state and therefore reduce the simulation time needed to reach a steady-state.

   Either YES or NO

   DEFAULT: NO

Notes

When this keyword is used ECLIPSE reads the data set up to the first timestepping keyword. The simulator is run up to a maximum amount of time specified in item 10 or it automatically stops when it believes, based on items 4-9, that a steady-state has been reached. SUMMARY section keywords PSSPR, PSSS(PSSSO, PSSSW, PSSSG) output the logarithm of the error and switch 77 of the DEBUG keyword outputs the actual errors. At the end of a successful run a restart file is produced (usually number 1 but check the print file to be sure) and this restart file is dated with the date given in items 1-3 regardless of what time the simulator is reporting as being current.

While timestepping keywords are ignored the SCHEDULE section is read, and honored, up to the first timestepping keyword. This means that steady-states can be generated with wells producing. However, steady-states may not exist in all cases. If wells are changing controls a steady-state will not be achieved. If it is desired to have a steady state with a well that is open, but the well can only flow near the steady-state solution then there is a problem as the well cannot be opened later in the SCHEDULE section as no more timestep keywords will be read. There are two options. The keyword option means ensuring that the well is tested for opening by using the WTEST keyword, or the ACTIONX keyword can be used to open the well at a date sufficiently later that it can operate. item 11 can be helpful to ensure the code doesn’t end before the ACTION is triggered.

A restart file will be produced in all cases so it is necessary to check in the print file that steady-state was successfully accomplished.

Note: PSTEADY cannot be used in a restart run or a parallel run.

Example

```
PSTEADY 1 JAN 1980 MAXABS 5* 5000.0 /
```
Primary saturation function region numbers

The keyword should be followed by one integer for every grid block in the current input box specifying the primary saturation function region to which it belongs. This keyword is used together with ISTNUM and DSTNUM if the ODD3P relative permeability and capillary pressure hysteresis model has been selected in the PROPS section. The region number should not be less than 1 or greater than NTSFUN (see keyword TABDIMS). The data must be terminated by a slash (/).

The primary saturation function region number specifies which set of saturation functions (input using SOF3, SGF3, and SWF3 in the PROPS section) should be used to calculate the primary relative permeabilities and capillary pressures in each grid block.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

For grid blocks for which the primary saturation function region numbers are not specified, the region numbers will be assigned a value of 1. The ODD3P model is implicitly hysteretic. However, hysteresis may be deselected on a per grid block basis by setting the secondary and tertiary hysteresis saturation region numbers equal to the primary saturation region numbers.

See also the saturation function keywords SOF3, SGF3 and SWF3 in the PROPS section and the secondary and tertiary hysteresis saturation region numbers ISTNUM and DSTNUM in the REGIONS section.

Example

With NTSFUN=2 specified in keyword TABDIMS; NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS; and no input BOX set:

```
PSTNUM
144*1 96*2 /
```
Principal stress boundary conditions coupled stress and fluid flow

This keyword is being replaced by a new keyword for setting stress boundary conditions. Please refer to keyword GMPSTBC.

This keyword specifies principal stress boundary conditions when a rock stress calculation is specified. It is only applicable when the keyword GEOMECH has been specified in the RUNSPEC section.

Use of this keyword allows the full set of three principal stress vectors to be specified on all boundaries. This is sufficient information to fully define the stress tensor at the chosen boundary. Alternatively, tractions or displacements may be specified using the TRACTBC or DISPBC keywords.

Note that the principal stress vectors should, by definition, be mutually orthogonal. If non-orthogonal vectors are provided, ECLIPSE 300 issues a warning message and orthogonalizes the vectors provided using the Gram-Schmidt process.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the principal stress boundary condition is applied
   This must be one of the following:
   - I+  Specify the boundary condition on the outer I+ edge
   - I−  Specify the boundary condition on the outer I- edge
   - J+  Specify the boundary condition on the outer J+ edge
   - J−  Specify the boundary condition on the outer J- edge
   - K+  Specify the boundary condition on the outer K+ edge
   - K−  Specify the boundary condition on the outer K- edge

   DEFAULT: There is no default for this item; it must be specified.
   The K- boundary is the top boundary of the simulation, K+ is the bottom.

2. The X component of the first principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

3. The Y component of the first principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0
4. The Z component of the first principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

5. The X component of the second principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

6. The Y component of the second principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

7. The Z component of the second principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

8. The X component of the third principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

10. The Z component of the third principal stress vector.
    UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
        DEFAULT: 0

11. First block to be modified in the I direction
    DEFAULT: 1 if an I- face

12. Last block to be modified in the I direction
    DEFAULT: NX if an I+ face

13. First block to be modified in the J direction
    DEFAULT: 1 if a J- face

14. Last block to be modified in the J direction
    DEFAULT: NY if a J+ face

15. First block to be modified in the K direction
    DEFAULT: 1 if a K- face

16. Last block to be modified in the K direction
    DEFAULT: NZ if a K+ face
### Example

Principal stress vectors:
- \( P_1 = (-2000, 0, 0) \), \( P_2 = (0, -2500, 200) \), \( P_3 = (0, 0, 0) \)
- are defined on all boundary faces of the entire grid.

<table>
<thead>
<tr>
<th>Side</th>
<th>( P_{1x} )</th>
<th>( P_{1y} )</th>
<th>( P_{1z} )</th>
<th>( P_{2x} )</th>
<th>( P_{2y} )</th>
<th>( P_{2z} )</th>
<th>( P_{3x} )</th>
<th>( P_{3y} )</th>
<th>( P_{3z} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>'I-'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'I+'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'J-'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'J+'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'K-'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'K+'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
</tbody>
</table>
Principal stress boundary conditions coupled stress and fluid flow in a Local Grid Refinement

This keyword is being replaced by new keyword for setting displacement boundary conditions. Please refer to keyword **GMPSTBCL**.

This keyword specifies principal stress boundary conditions when a finite volume or finite element rock stress calculation is specified. It is only applicable when the keyword **GEOMECH** has been specified in the RUNSPEC section.

Use of this keyword allows the full set of three principal stress vectors to be specified on all boundaries in a local grid refinement. This is sufficient information to fully define the stress tensor at the chosen boundary. Alternatively, tractions or displacements may be specified using the **TRACTBCL** or **DISPBCL** keywords.

Note that the principal stress vectors should, by definition, be mutually orthogonal. If non-orthogonal vectors are provided ECLIPSE 300 issues a warning message and orthogonalizes the vectors provided using the Gram-Schmidt process.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the principal stress boundary condition is applied
   This must be one of the following:
   - **I+**: Specify the boundary condition on the outer I+ edge
   - **I-**: Specify the boundary condition on the outer I- edge
   - **J+**: Specify the boundary condition on the outer J+ edge
   - **J-**: Specify the boundary condition on the outer J- edge
   - **K+**: Specify the boundary condition on the outer K+ edge
   - **K-**: Specify the boundary condition on the outer K- edge
   
   **DEFAULT**: There is no default for this item; it must be specified.

   The K- boundary is the top boundary of the simulation, K+ is the bottom.

   Name of local grid refinement

2. The X component of the first principal stress vector.
   **UNITS**: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   **DEFAULT**: 0

3. The Y component of the first principal stress vector.
   **UNITS**: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
4. The Z component of the first principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

5. The X component of the second principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

6. The Y component of the second principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

7. The Z component of the second principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

8. The X component of the third principal stress vector.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

10. The Z component of the third principal stress vector.
    UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
       DEFAULT: 0

11. First block to be modified in the I direction
    DEFAULT: 1 if an I- face

12. Last block to be modified in the I direction
    DEFAULT: NX if an I+ face

13. First block to be modified in the J direction
    DEFAULT: 1 if a J- face

14. Last block to be modified in the J direction
    DEFAULT: NY if a J+ face

15. First block to be modified in the K direction
    DEFAULT: 1 if a K- face

16. Last block to be modified in the K direction

Keywords

1744
DEFUALT: NZ if a K+ face

Example

-- Principal stress vectors:
-- P1=(-2000,0,0), P2=(0,-2500,200), P3=(0,0,0)
-- are defined on all boundary faces of the local grid refinement LG1.

<table>
<thead>
<tr>
<th>side</th>
<th>LGR</th>
<th>P1X</th>
<th>P1Y</th>
<th>P1Z</th>
<th>P2X</th>
<th>P2Y</th>
<th>P2Z</th>
<th>P3X</th>
<th>P3Y</th>
<th>P3Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>'I-'</td>
<td>'LG1'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'I+'</td>
<td>'LG1'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'J-'</td>
<td>'LG1'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'J+'</td>
<td>'LG1'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'K-'</td>
<td>'LG1'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>'K+'</td>
<td>'LG1'</td>
<td>-2000</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>-2500</td>
<td>200</td>
<td>9*</td>
<td>/</td>
<td></td>
</tr>
</tbody>
</table>

/
PSWRG  Primary residual water saturation in water-gas system

This keyword specifies the residual water saturation in a water-gas system within each grid cell for the primary saturation tables. This is only required when the ODD3P option is being used and when saturation table end-point scaling has been specified by the ENDScale keyword in the RUNSPEC section.

Each PSWRG keyword should be followed by one real number for each grid cell in the current input box specifying the residual saturation within the grid cell. The saturation should be in the range of 0.0 to 1.0 inclusive. If the value of PSWRG is omitted for any grid cell in a model for which ENDScale has been specified, the residual saturation will be obtained from the critical saturation in the appropriate saturation table.

See "Saturation table scaling" in the ECLIPSE Technical Description for further details.

See also the PSORG, PSGRO, PSORW, PSWRO and PSGRW keywords.

Example

With NDIVIX=3, NDIVIY=4 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS and no input BOX set:

```
PSWRG
60*0.1/
```

Keywords

1746
PSWRO

Primary residual water saturation in water-oil system

This keyword specifies the residual water saturation in a water-oil system within each grid cell for the primary saturation tables. This is only required when the ODD3P option is being used and when saturation table end-point scaling has been specified by the ENDSCALE keyword in the RUNSPEC section.

Each PSWRO keyword should be followed by one real number for each grid cell in the current input box specifying the residual saturation within the grid cell. The saturation should be in the range of 0.0 to 1.0 inclusive. If the value of PSWRO is omitted for any grid cell in a model for which ENDScale has been specified, the residual saturation will be obtained from the critical saturation in the appropriate saturation table.

See "Saturation table scaling" in the ECLIPSE Technical Description for further details.

See also the PSORG, PSGRO, PSORW, PSGRW and PSWRG keywords.

Example

With NDIVIX=3, NDIVIY=4 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS and no input BOX set:

```
PSWRO
60*0.1/
```
PVCDO

Dead oil PVT properties (with constant compressibility)

The data comprises NTPVT (see item 2 of keyword TABDIMS in the RUNSPEC section) records, one for each PVT table. Each record must be terminated by a slash (/).

Each record consists of the following items of data:

1. The reference pressure \( P_{\text{ref}} \) for items 2 and 4.
   
   **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2. The oil formation volume factor at the reference pressure, \( B_o(P_{\text{ref}}) \)
   
   **UNITS:** \( \text{m}^3/\text{sm}^3 \) (METRIC), rb/stb (FIELD), rcc/scc (LAB), \( \text{m}^3/\text{sm}^3 \) (PVT-M)

3. The oil compressibility

\[
C = \left( \frac{dB_o}{dP} \right) B_o
\]

**UNITS:** 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

4. The oil viscosity at the reference pressure \( \mu_o(P_{\text{ref}}) \)

**UNITS:** cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

5. The oil “viscosibility”

\[
C_v = \left( \frac{d\mu_o}{dP} \right) \mu_o
\]

**UNITS:** 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

ECLIPSE calculates \( B_o \) using:

\[
B_o(P) = B_o(P_{\text{ref}}) e^{-X}
\]

Eq. 3.150

where

\[ X = C(P-P_{\text{ref}}) \]

and the product \( B_o\mu_o \) using:

\[
B_o\mu_o P = B_o P_{\text{ref}} \mu_o P_{\text{ref}} e^{-Y}
\]

Eq. 3.151

where

\[ Y = (C-C_v)(P-P_{\text{ref}}) \]

See also keyword PVDO.

**Example**

With NTPVT=1:

---

**Keywords**

1748
PVCDO
3600 1.12 1.6e-5 0.88 0.0 /
PVC0

PVT properties of live oil in compressibility form (with dissolved gas)

This keyword offers a simpler method of entering tables of live oil PVT properties than with the PVTO keyword. The difference between the two keywords is that the PVC0 keyword assumes that undersaturated oil with a particular $R_s$ value has a compressibility that is independent of the pressure, and that the viscosity of undersaturated oil has a pressure-independent derivative. Thus there is no need to provide tables of undersaturated oil formation volume factor and viscosity versus pressure.

The data comprises NTPVT (see keyword TABDIMS in the RUNSPEC section) tables of live oil PVT functions. A single table consists of at least 2 and up to NRPVT (keyword TABDIMS) rows of 6 columns. Each table is terminated by a slash (/).

The columns are:

1. The bubble point pressure ($P_{bub}$) for oil with the dissolved gas-oil ratio given by column 2.
   Within each table, the pressure values should increase monotonically down the column.
   UNITS: bars (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)

2. The dissolved gas-oil ratio ($R_s$) of saturated oil with bubble point given by column 1.
   The values should increase monotonically down the column.
   UNITS: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)

3. The formation volume factor of saturated oil at $P_{bub}$.
   UNITS: rm$^3$/sm$^3$ (METRIC), rb/stb (FIELD), rcc/scc (LAB), rm$^3$/sm$^3$ (PVT-M)

4. The viscosity of saturated oil at $P_{bub}$.
   UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

5. The compressibility of undersaturated oil with a dissolved gas-oil ratio given by column 2:
   \[ C = -\left(\frac{dB_o}{dP}\right)\Big|_{B_o} \]
   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

6. The “viscosibility”, or viscosity compressibility, of undersaturated oil with a dissolved gas-oil ratio given by column 2:
   \[ C_v = \left(\frac{d\mu_o}{dP}\right)\Big|_{\mu_o} \]
   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

Note: When the PVC0 keyword is used within a thermal model to set the oil viscosity pressure dependence, only columns 1 and 4 need to be entered. Other columns are not used and should be defaulted. See “Thermal option” in the ECLIPSE Technical Description.

The sixth column can be ignored by setting item 24 in OPTIONS3 to -1. It may be left out entirely if item 24 in OPTIONS3 is set > 0, which restores the pre-99A five-column format of this keyword.
There must be the same number of entries in each column of a given table. This should not be less than 2 or greater than \(\text{NRPVT}\) (keyword `TABDIMS`).

Default values (represented by 1*) may be inserted as required in columns 2 to 6. When a table is read in, defaults are replaced by values computed by linear interpolation. To input a constant compressibility (or viscosibility) for the whole table, enter the value of this quantity in the first row and enter defaults for the remaining items in the column. If all the items in the viscosibility column are defaulted, a value of zero will be assumed.

If the `PVCO` keyword is used, the maximum pressure expected in the simulation must also be entered with keyword `PMAX`. This information is used for two purposes. Firstly, the undersaturated parts of the oil PVT tables are automatically filled out in the form of the `PVTO` keyword, up to the pressure entered in `PMAX`.

**Note:** The undersaturated parts of the oil PVT tables are extrapolated over \(\text{NPPVT}\) pressure nodes (see item 4 of keyword `TABDIMS` in the `RUNSPEC` section) between the bubble point pressure and the maximum pressure entered in `PMAX`. This means that interpolation between pressure nodes will vary slightly depending on the number of nodes used and will affect the simulation results accordingly. To avoid extrapolation, the keyword `PVTO` should be used instead of `PVCO`.

Secondly, ECLIPSE carries out a check to see if compressibilities are changing too rapidly with \(P_b\) so that the curves of undersaturated oil formation volume factor for two different values of \(R_s\) cross each other. A warning is printed if they cross at a pressure below the maximum pressure entered in `PMAX`.

**Note:** ECLIPSE 100 linearly interpolates the reciprocals of \(B_o\) and \((B_o\mu_o)\) between data points, rather than the values themselves. This should be taken into account when comparing the results of ECLIPSE with those of other simulators, by ensuring the data points are not distributed too sparsely.

**Note:** ECLIPSE 300 linearly interpolates both \(B_o\) and \(\mu_o\) directly. If item 56 of the `OPTIONS3` keyword is set greater than 0, then `PVCO` data will be interpolated in the same way as ECLIPSE 100.

**Note:** When using the API Tracking option (keyword `API` in the `RUNSPEC` section), the set of tables must be entered in increasing order of surface density (or decreasing order of API gravity).

See also keyword `PVTO`.

**Example**

With \(\text{NTPVT}=2\), \(\text{NRPVT}\) must have been set \(\geq 14\):

<table>
<thead>
<tr>
<th>PVCO</th>
<th>0.165</th>
<th>1.012</th>
<th>1.17</th>
<th>1.0E-5</th>
<th>1*</th>
</tr>
</thead>
<tbody>
<tr>
<td>400</td>
<td>0.165</td>
<td>1.012</td>
<td>1.17</td>
<td>1.0E-5</td>
<td>1*</td>
</tr>
<tr>
<td>800</td>
<td>0.335</td>
<td>1*</td>
<td>1.14</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>1200</td>
<td>0.500</td>
<td>1.038</td>
<td>1*</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>1600</td>
<td>0.665</td>
<td>1.051</td>
<td>1.08</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>2000</td>
<td>0.828</td>
<td>1*</td>
<td>1.06</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>2400</td>
<td>0.985</td>
<td>1.075</td>
<td>1*</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>2800</td>
<td>1.130</td>
<td>1.087</td>
<td>1.00</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>3200</td>
<td>1.270</td>
<td>1*</td>
<td>0.98</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>3600</td>
<td>1.390</td>
<td>1.11</td>
<td>1*</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>4000</td>
<td>1.500</td>
<td>1.12</td>
<td>0.94</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>4400</td>
<td>1.600</td>
<td>1*</td>
<td>0.92</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>Value</td>
<td>1.676</td>
<td>1.14</td>
<td>1*</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>--------</td>
<td>-------</td>
<td>------</td>
<td>-----</td>
<td>-----</td>
<td>-----</td>
</tr>
<tr>
<td>5200</td>
<td>1.750</td>
<td>1.148</td>
<td>0.9</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>5600</td>
<td>1.810</td>
<td>1.155</td>
<td>0.89</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>400</td>
<td>0.18</td>
<td>1.02</td>
<td>1.12</td>
<td>1.0E-5</td>
<td>1*</td>
</tr>
<tr>
<td>1200</td>
<td>0.520</td>
<td>1.042</td>
<td>1.07</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>2000</td>
<td>0.84</td>
<td>1.070</td>
<td>1.01</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>2800</td>
<td>1.200</td>
<td>1.091</td>
<td>0.94</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>3600</td>
<td>1.50</td>
<td>1.124</td>
<td>0.90</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>4400</td>
<td>1.720</td>
<td>1.146</td>
<td>0.88</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>5200</td>
<td>1.870</td>
<td>1.161</td>
<td>0.87</td>
<td>1*</td>
<td>1*</td>
</tr>
<tr>
<td>5600</td>
<td>1.990</td>
<td>1.172</td>
<td>0.86</td>
<td>2.0E-5</td>
<td>1*</td>
</tr>
</tbody>
</table>

**Keywords**

1752
PVT properties of dry gas (no vaporized oil)

The data comprises NTPVT (item 2 in keyword \texttt{TABDIMS} in the \texttt{RUNSPEC} section) tables of dry gas PVT functions, each terminated by a slash (/).

Each table consists of 3 columns of data:

1. The gas phase pressure.
   
   Values should increase monotonically down the column.
   
   \noindent UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2. The corresponding gas formation volume factor.
   
   Values should decrease down the column.
   
   \noindent UNITS: \text{rm$^3$/sm$^3$} (METRIC), rb/Mscf (FIELD), rcc/scc (LAB), \text{rm$^3$/sm$^3$} (PVT-M)

3. The corresponding gas viscosity.
   
   Values should be level or increasing down the column.
   
   \noindent UNITS: \text{cP} (METRIC), \text{cP} (FIELD), \text{cP} (LAB), \text{cP} (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than \texttt{NPPVT} (item 4 in keyword \texttt{TABDIMS}).

See also keyword \texttt{PVZG}.

\textit{Note:} ECLIPSE 100 linearly interpolates the reciprocals of $B_g$ and $(B_g \mu_g)$ between data points, rather than the values themselves. This should be taken into account when comparing the results of ECLIPSE with those of other simulators, by ensuring the data points are not distributed too sparsely.

\textit{Note:} ECLIPSE 300 linearly interpolates both $B_g$ and $\mu_g$ directly. If item 56 of the \texttt{OPTIONS3} keyword is set to 2, then \texttt{PVDG} data (as well as oil property data) will be interpolated in the same way as ECLIPSE 100.

\textbf{Example}

With one pressure table:

\begin{verbatim}
PVDG
  400.000 5.90000 .01300
  800.000 2.95000 .01350
 1200.00 1.96000 .01400
 1600.00 1.47000 .01450
 2000.00 1.18000 .01500
 2400.00 .98000 .01550
 2800.00 .84000 .01600
 3200.00 .74000 .01650
 3600.00 .65000 .01700
 4000.00 .59000 .01750
 4400.00 .54000 .01800
 4800.00 .49000 .01850
 5200.00 .45000 .01900
 5600.00 .42000 .01950
\end{verbatim}
PVDO

PVT properties of dead oil (no dissolved gas)

The data comprises NTPVT (see item 2 of keyword TABDIMS in the RUNSPEC section) tables of dead oil PVT functions, each terminated by a slash (/).

Each table consists of 3 columns of data:

1. The oil phase pressure.
   - Values should increase monotonically down the column.
   - UNITS: barsa (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)

2. The corresponding oil formation volume factor.
   - Values should decrease down the column.
   - UNITS: m³/sm³ (METRIC), rb/stb (FIELD), rcc/scc (LAB), m³/sm³ (PVT-M)

3. The corresponding oil viscosity.
   - Values should be level or increasing down the column.
   - UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (item 4 of keyword TABDIMS).

See also keyword PVCD0.

**ECLIPSE 100**

*Note:* ECLIPSE 100 linearly interpolates the reciprocals of $B_o$ and $(B_o \mu_o)$ between data points, rather than the values themselves. This should be taken into account when comparing the results of ECLIPSE with those of other simulators, by ensuring the data points are not distributed too sparsely.

**ECLIPSE 300**

*Note:* ECLIPSE 300 linearly interpolates both $B_o$ and $\mu_o$ directly. If item 56 of the OPTIONS3 keyword is set greater than 0, then PVDO data will be interpolated in the same way as ECLIPSE 100.

**Example**

With NTPVT=2 and NPPVT≥8:

<table>
<thead>
<tr>
<th>PVDO</th>
<th>400</th>
<th>1.012</th>
<th>1.16</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1200</td>
<td>1.0040</td>
<td>1.164</td>
</tr>
<tr>
<td></td>
<td>2000</td>
<td>0.9960</td>
<td>1.167</td>
</tr>
<tr>
<td></td>
<td>2800</td>
<td>0.9880</td>
<td>1.172</td>
</tr>
<tr>
<td></td>
<td>3600</td>
<td>0.9802</td>
<td>1.177</td>
</tr>
<tr>
<td></td>
<td>4400</td>
<td>0.9724</td>
<td>1.181</td>
</tr>
<tr>
<td></td>
<td>5200</td>
<td>0.9646</td>
<td>1.185</td>
</tr>
<tr>
<td></td>
<td>5600</td>
<td>0.9607</td>
<td>1.19</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>1.0255</td>
<td>1.14</td>
</tr>
<tr>
<td></td>
<td>1600</td>
<td>1.0172</td>
<td>1.14</td>
</tr>
<tr>
<td></td>
<td>2400</td>
<td>1.0091</td>
<td>1.14</td>
</tr>
<tr>
<td></td>
<td>3200</td>
<td>1.0011</td>
<td>1.14</td>
</tr>
<tr>
<td></td>
<td>4000</td>
<td>0.9931</td>
<td>1.14</td>
</tr>
<tr>
<td></td>
<td>4800</td>
<td>0.9852</td>
<td>1.14</td>
</tr>
<tr>
<td></td>
<td>5600</td>
<td>0.9774</td>
<td>1.14</td>
</tr>
</tbody>
</table>
PVDS

PVT properties of the solvent

This keyword may be used with the solvent or coal bed methane model as described in "Solvent model" and "Coal bed methane model" in the ECLIPSE Technical Description. The solvent is a second gas phase within the reservoir. The PVT data required is essentially a second dry gas table, analogous to the PVDG table.

The data comprises NTPVT (see keyword TABDIMS) tables of solvent PVT functions, each terminated by a slash (/). Each table consists of 3 columns of data:

1. The solvent gas phase pressure.
   Values should increase monotonically down the column.
   UNITS: bara (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)

2. The corresponding solvent formation volume factor.
   Values should decrease down the column.
   UNITS: rm³/sm³ (METRIC), rb/Mscf (FIELD), rcc/scc (LAB), rm³/sm³ (PVT-M)

3. The corresponding solvent viscosity.
   Values should be level or increasing down the column.
   UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see keyword TABDIMS).

Example

With NTPVT=1 and NPPVT≥14

```
--  PGAS  BGAS  VISGAS
PVDS
400 5.9  0.013
800 2.95 0.0135
1200 1.96 0.014
1600 1.47 0.0145
2000 1.18 0.015
2400 0.98 0.0155
2800 0.84 0.016
3200 0.74 0.0165
3600 0.65 0.017
4000 0.59 0.0175
4400 0.54 0.018
4800 0.49 0.0185
5200 0.45 0.019
5600 0.42 0.0195 /
```
PVTG

PVT properties of wet gas (with vaporized oil)

The data comprises NTPVT (see item 2 in keyword TABDIMS in the RUNSPEC section) tables of wet gas PVT functions. A single table consists of at least 2 and up to NPPVT (item 4 in keyword TABDIMS) records each terminated by a slash (/). Each record gives PVT data for a particular gas phase pressure (P_g).

A table is terminated by a null record (that is a record with no data before the terminating slash). This table may be generated by PVT i.

Most records contain the following items of data and a terminating slash:

1. The gas phase pressure (P_g).
   
   Within a table, records should be arranged in order of increasing P_g.
   
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atm (PVT-M)

2. The vaporized oil-gas ratio for saturated gas at pressure P_g.
   
   UNITS: sm^3/sm^3 (METRIC), stb/Mscf (FIELD), scc/scc (LAB), sm^3/sm^3 (PVT-M)

3. The gas formation volume factor for saturated gas at P_g.
   
   UNITS: rm^3/sm^3 (METRIC), rb/Mscf (FIELD), rcc/scc (LAB), rm^3/sm^3 (PVT-M)

4. The gas viscosity for saturated gas at P_g.
   
   UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

However some records (optionally all) contain additional data that defines the properties of under-saturated gas at the specified value of P_g. This extra data must be specified for the highest P_g in each table. The additional data takes the form of 3 columns which continue from items 2, 3 and 4 above.

Each column may have up to NRPVT (item 6 in keyword TABDIMS) entries.

Column 2 The vaporized oil-gas ratio (R_v).

Values of R_v must decrease down the column (starting from the value for saturated gas).

   UNITS: sm^3/sm^3 (METRIC), stb/Mscf (FIELD), scc/scc (LAB), sm^3/sm^3 (PVT-M)

Column 3 The gas formation volume factor (B_g) corresponding to R_v and P_g.

   UNITS: rm^3/sm^3 (METRIC), rb/Mscf (FIELD), rcc/scc (LAB), rm^3/sm^3 (PVT-M)

Column 4 The gas viscosity (\mu_g) corresponding to R_v and P_g.

   UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

There must be the same number of entries in each column. If no additional data is supplied for a particular value of P_g, ECLIPSE locates the next higher value of P_g for which undersaturated PVT data is available, and takes a scaled copy of the FVF and viscosity curves supplied there.

Notes

- Data for undersaturated gas must be provided for the highest P_g in the table.

ECLIPSE 100

Keywords

1756
• ECLIPSE 100 linearly interpolates the **reciprocals** of $B_g$ and $(B_g \mu_g)$ between data points, rather than the values themselves. This should be taken into account when comparing the results of ECLIPSE 100 with those of other simulators, by ensuring the data points are not distributed too sparsely.

• ECLIPSE 300 linearly interpolates the **reciprocals** of $B_g$ but $\mu_g$ is interpolated directly.

**Example**

With NTPVT=2, NRPVT must have been set ≥2 and NPPVT≥7:

```plaintext
<table>
<thead>
<tr>
<th>PVTG</th>
<th>0.00014</th>
<th>0.0523</th>
<th>0.0234</th>
</tr>
</thead>
<tbody>
<tr>
<td>30</td>
<td>0.00012</td>
<td>0.0132</td>
<td>0.0252</td>
</tr>
<tr>
<td>90</td>
<td>0.00015</td>
<td>0.00477</td>
<td>0.0281</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PVTG</th>
<th>0.00015</th>
<th>0.00877</th>
<th>0.0275</th>
</tr>
</thead>
<tbody>
<tr>
<td>150</td>
<td>0.00012</td>
<td>0.00554</td>
<td>0.0318</td>
</tr>
<tr>
<td>210</td>
<td>0.00015</td>
<td>0.00417</td>
<td>0.0355</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PVTG</th>
<th>0.00015</th>
<th>0.00877</th>
<th>0.0275</th>
</tr>
</thead>
<tbody>
<tr>
<td>150</td>
<td>0.00012</td>
<td>0.00554</td>
<td>0.0318</td>
</tr>
<tr>
<td>210</td>
<td>0.00015</td>
<td>0.00417</td>
<td>0.0355</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PVTG</th>
<th>0.00014</th>
<th>0.00877</th>
<th>0.0275</th>
</tr>
</thead>
<tbody>
<tr>
<td>150</td>
<td>0.00012</td>
<td>0.00554</td>
<td>0.0318</td>
</tr>
<tr>
<td>210</td>
<td>0.00015</td>
<td>0.00417</td>
<td>0.0355</td>
</tr>
</tbody>
</table>

/ null record to terminate table 1

<table>
<thead>
<tr>
<th>PVTG</th>
<th>0.00014</th>
<th>0.0523</th>
<th>0.0234</th>
</tr>
</thead>
<tbody>
<tr>
<td>30</td>
<td>0.00012</td>
<td>0.0132</td>
<td>0.0252</td>
</tr>
<tr>
<td>90</td>
<td>0.00015</td>
<td>0.00477</td>
<td>0.0281</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PVTG</th>
<th>0.00015</th>
<th>0.00877</th>
<th>0.0275</th>
</tr>
</thead>
<tbody>
<tr>
<td>150</td>
<td>0.00012</td>
<td>0.00554</td>
<td>0.0318</td>
</tr>
<tr>
<td>210</td>
<td>0.00015</td>
<td>0.00417</td>
<td>0.0355</td>
</tr>
</tbody>
</table>

/ null record to terminate table 1

<table>
<thead>
<tr>
<th>PVTG</th>
<th>0.00014</th>
<th>0.00877</th>
<th>0.0275</th>
</tr>
</thead>
<tbody>
<tr>
<td>150</td>
<td>0.00012</td>
<td>0.00554</td>
<td>0.0318</td>
</tr>
<tr>
<td>210</td>
<td>0.00015</td>
<td>0.00417</td>
<td>0.0355</td>
</tr>
</tbody>
</table>

/ null record to terminate table 1

null record to terminate table 2
```

Keywords
PVT-M

**PVT-metric units are to be used**

This keyword specifies that PVT-metric units are to be used for the input data. These are essentially the same as METRIC units, but with pressures measured in atmospheres. Note that the default units are METRIC.

See "Units" in the ECLIPSE Technical Description for further details.

There is no data associated with this keyword.
**PVTNUM**

**PVT region numbers**

The keyword should be followed by one integer for every grid block in the current input box specifying the PVT region to which it belongs. The region number should not be less than 1 or greater than NTPVT (item 2 in keyword **TABDIMS**). All grid blocks within a particular equilibration region (see keyword **EQLNUM**) must have the same PVT region number. The data must be terminated by a slash (/).

The PVT region number specifies which set of PVT tables (input using `DENSITY`, `PVDG`, `PVDO`, `PVTG`, `PVTO`, `PVCO`, `PVTw` and `ROCK` in the **PROPS** section) should be used to calculate PVT properties of fluids in each grid block for a black oil model.

For a compositional model, `PVTNUM` is only used to reference the water properties defined via `DENSITY`, `PVTw` or `WATERTAB` and the rock compressibility specified by `ROCK` if required.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

**Example**

With NTPVT=4 specified in keyword **TABDIMS**; NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in **RUNSPEC** keyword **DIMENS**; and no input **BOX** set:

```
PVTNUM
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4 /
```
PVT properties of live oil (with dissolved gas)

The data comprises NTPVT (see item 2 of keyword TABDIMS in the RUNSPEC section) tables of live oil PVT functions. A single table consists of at least 2 and up to NRPVT (item 6 of keyword TABDIMS) records each terminated by a slash (/). Each record gives PVT data for a particular dissolved gas-oil ratio (R_s). A table is terminated by a null record (that is a record with no data before the terminating slash).

Most records contain the following items of data and a terminating slash:

1. The dissolved gas-oil ratio (R_s).
   
   Within a table, records should be arranged in order of increasing R_s.
   
   **UNITS:** sm^3/sm^3 (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm^3/sm^3 (PVT-M)

2. The bubble point pressure (P_bub) for oil with dissolved gas-oil ratio given by item 1.
   
   **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

3. The oil formation volume factor for saturated oil at P_bub.
   
   **UNITS:** rm^3/sm^3 (METRIC), rb/stb (FIELD), rcc/scc (LAB), rm^3/sm^3 (PVT-M)

4. The oil viscosity for saturated oil at P_bub.
   
   **UNITS:** cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

However some records (optionally all) contain additional data which defines the properties of undersaturated oil at the specified value of R_s. This extra data must be specified for the highest R_s in each table. The additional data takes the form of 3 columns that continue from items 2, 3 and 4 above. Each column may have up to NPPVT (see item 4 in keyword TABDIMS) entries.

**Column 2** The oil phase pressure (P_o). Values of P_o must increase down the column (starting from P_bub).

**UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

**Column 3** The oil formation volume factor (B_o) corresponding to R_s and P_o. Values of B_o must decrease down the column.

**UNITS:** rm^3/sm^3 (METRIC), rb/stb (FIELD), rcc/scc (LAB), rm^3/sm^3 (PVT-M)

**Column 4** The oil viscosity (μ_o) corresponding to R_s and P_o. Values of μ_o do not normally decrease down the column.

**UNITS:** cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

There must be the same number of entries in each column. If no additional data is supplied for a particular value of R_s, ECLIPSE locates the next higher value of R_s for which undersaturated PVT data is available, and takes a scaled copy of the FVF and viscosity curves supplied there. The scaling is done in such a way that the compressibility and ‘viscosibility’,

\[
\left(\frac{d\mu_o}{dP_o}\right)\bigg|_{\mu_o} \quad \text{Eq. 3.152}
\]

are preserved.

Note that data for undersaturated oil must be provided for the highest R_s in the table.
Notes

**ECLIPSE 100**
- ECLIPSE 100 linearly interpolates the **reciprocals** of $B_0$ and $(B_0 \mu_0)$ between data points, rather than the values themselves. This should be taken into account when comparing the results of ECLIPSE 100 with those of other simulators, by ensuring the data points are not distributed too sparsely.

**ECLIPSE 300**
- ECLIPSE 300 linearly interpolates the **reciprocals** of $B_0$ but $\mu_o$ is interpolated directly. If item 56 of the **OPTIONS3** keyword is set greater than 0, then $PVTO$ data will be interpolated in the same way as ECLIPSE 100.
- When using the API Tracking option (keyword **API** in the **RUNSPEC** section), the set of tables must be entered in increasing order of surface density (or decreasing order of API gravity).

See also keyword **PVCO**.

Example

With **NTPVT**=2. **NRPVT** must have been set $\geq 11$ and **NPPVT**$\geq 6$:

```plaintext
PVTO
0.165  400  1.012   1.17  /
0.335  800  1.0255  1.14  /
0.500  1200 1.038  1.11  /
0.665  1600 1.051  1.08  /
0.828  2000 1.063  1.06  /
0.985  2400 1.075  1.03  /
1.130  2800 1.087  1.00  /
1.270  3200 1.0985 0.98  /
1.390  3600 1.11  0.95  /
1.500  4000 1.12  0.94  /
1.600  4400 1.13  0.92  /
   4800 1.1255 0.92  /
   5200 1.1210 0.92  /
   5600 1.1165 0.92  / data for undersaturated oil with Rs=1.6
/ null record terminates table 1
0.275  400  1.1334  1.17  /
0.610  1200 1.1626  1.11  /
0.938  2000 1.1906  1.06  /
1.240  2800 1.2174  1.00  /
1.500  3600 1.2432  0.95  /
4000  1.2382  0.95  /
   4400 1.2332  0.95  /
   4800 1.2283  0.95  /
   5200 1.2235  0.95  /
   5600 1.2186  0.95  / data for undersaturated oil with Rs=1.5
1.610  4000 1.2544  0.94  /
1.720  4400 1.2656  0.92  /
   4800 1.2606  0.92  /
   5200 1.2555  0.92  /
   5600 1.2505  0.92  / data for undersaturated oil with Rs=1.72
/ null record terminates table 2
```
Water PVT functions

The data comprises NTPVT (see item 2 of keyword TABDIMS in the RUNSPEC section) records, each terminated by a slash (/).

Each record consists of the following items of data:

1. The reference pressure ($P_{\text{ref}}$) for items 2 and 4.
   - UNITS: barsa (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)
   - DEFAULT: ERROR (ECLIPSE 100), 1 atm = 14.7 psia (ECLIPSE 300).

2. The water formation volume factor at the reference pressure, $B_w(P_{\text{ref}})$
   - UNITS: $\text{rm}^3/\text{sm}^3$ (METRIC), $\text{rb/stb}$ (FIELD), $\text{rcc/scc}$ (LAB), $\text{rm}^3/\text{sm}^3$ (PVT-M)
   - DEFAULT: 1.0 (in all units)

3. The water compressibility
   \[ C = \left( \frac{dB_w}{dP} \right) B_w \]  
   - UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)
   - DEFAULT: 0 (ECLIPSE 100), 4.0e-5 1/atm (ECLIPSE 300)

4. The water viscosity at the reference pressure $\mu_w(P_{\text{ref}})$
   - UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)
   - DEFAULT: 0.5 cP (ECLIPSE 100), 0.3 cP (ECLIPSE 300).

5. The water “viscosibility”
   \[ C_v = \left( \frac{d\mu_w}{dP} \right) \mu_w \]  
   - UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)
   - DEFAULT: 0.0

**ECLIPSE 100**

ECLIPSE 100 calculates $B_w$ using:

\[ B_w(P) = \frac{B_w(P_{\text{ref}})}{1 + X + (X^2/2)} \]  

where

\[ X = C(P-P_{\text{ref}}) \]

ECLIPSE 100 calculates the product $B_w\mu_w$ using:

\[ B_w\mu_w(P) = \frac{B_w(P_{\text{ref}})\mu_w(P_{\text{ref}})}{1 + Y + (Y^2/2)} \]  

Keywords
where

\[ Y = (C - C_v)(P - P_{\text{ref}}) \]

**ECLIPSE 300**

ECLIPSE 300 calculates \( B_w \) using equation 3.155. \( \mu_w \) is calculated similarly:

\[ \mu_w(P) = \frac{\mu_w(P_{\text{ref}})}{1 + Y + (Y^2 / 2)} \quad \text{Eq. 3.157} \]

where

\[ Y = (C - C_v)(P - P_{\text{ref}}) \]

**ECLIPSE 300 Thermal option**

In the thermal option there are several ways of defining \( B_w \) and \( \mu_w \), which are described in "Water density" and "Water viscosity" in the ECLIPSE Technical Description. By default, most of the data given by this keyword is ignored.

**Example**

With NTPVT=2:

```
<table>
<thead>
<tr>
<th>PVTW</th>
<th>3600.0000</th>
<th>1.00341</th>
<th>3.00E-06</th>
<th>0.52341</th>
<th>0.00E-01</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>3900</td>
<td>1</td>
<td>2.67E-06</td>
<td>0.56341</td>
<td>1.20E-07</td>
</tr>
</tbody>
</table>
```

ECLIPSE Reference Manual

Keywords

1763
Water PVT functions with salt present

This keyword supplies the water PVT data for runs in which the Brine option is active (keyword BRINE). The keyword is used in place of the PVTW keyword.

The surface density of brine can be set for each PVTWSALT table using the BDENSITY keyword. This supplies surface densities at the values of salt concentration specified in the PVTWSALT keyword. If BDENSITY is missing, the surface density of brine is taken from the water surface density specified in DENSITY, for all salt concentrations.

In ECLIPSE 100 only, if the ECLMC keyword is selected, the BRINE keyword activates the multi-component brine model. In this case, salt effects on the water density are ignored and the keyword PVTW should be used instead.

In the present version of ECLIPSE 100 it is not possible to use this keyword with the polymer flood model. (See the PLYVISCs keyword for further information.)

The data comprises NTPVT (see keyword TABDIMS) tables. Each table consists of two records, each terminated by a slash (/).

Record 1
Two items of data:
1. The reference pressure ($P_{ref}$) for this table.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
2. The reference salt concentration for stock tank water, $c_{s \, ref}$.
   UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm$^3$ (PVT-M)
   DEFAULT: Minimum salt concentration specified in record 2 (typically zero).

Record 2
A table consisting of 5 columns of data.
The number of entries in the table should not exceed NPPVT (see keyword TABDIMS).

Note: In ECLIPSE 300 the number of entries must equal NPPVT. In ECLIPSE 100 a minimum of one entry is needed (which effectively gives constant properties).

1. The salt concentration, $c_s$.
   Values should increase monotonically down the column.
   UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm$^3$ (PVT-M)
2. The water formation volume factor at the reference pressure as a function of salt concentration, $B_w(P_{ref} \cdot c_s)$.
   Values should be level or decreasing down the column.
   UNITS: rm$^3$/sm$^3$ (METRIC), rb/stb (FIELD), rcc/scc (LAB), rm$^3$/sm$^3$ (PVT-M)
3. The water compressibility as a function of salt concentration,

\[ C(c_s) = -\left(\frac{dB_w}{dP}\right) B_w \]  

Eq. 3.158

UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

4. The water viscosity at the reference pressure as a function of salt concentration, \( \mu_w(P_{\text{ref}}, c_s) \).

UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

5. The water “viscosibility” as a function of salt concentration,

\[ C_v(c_s) = \left(\frac{d\mu_w}{dP}\right) \mu_w \]  

Eq. 3.159

UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

**ECLIPSE 100**

ECLIPSE first calculates the values of \( B_w, C, \mu_w, C_v \) as functions of the salt concentration at the previous timestep by linear interpolation in salt concentration for \( C \) and \( C_v \) and on the reciprocals of \( B_w \) and \( B_w\mu_w \). Then it calculates \( B_w \) and \( B_w\mu_w \) at the current pressure \( P \) as follows:

\[ B_w(P, c_s) = \frac{B_w(P_{\text{ref}}, c_s)}{1 + X + (X^2/2)} \]  

Eq. 3.160

where

\[ X = C(P-P_{\text{ref}}) \]

\[ B_w(P, c_s)\mu_w(P, c_s) = \frac{B_w(P_{\text{ref}}, c_s)\mu_w(P_{\text{ref}}, c_s)}{1 + Y + (Y^2/2)} \]  

Eq. 3.161

where

\[ Y = (C-C_v)(P-P_{\text{ref}}) \]

**ECLIPSE 300**

ECLIPSE 300 calculates \( B_w \) using equation 3.160. \( \mu_w \) is calculated similarly:

\[ \mu_w(P, c_s) = \frac{\mu_w(P_{\text{ref}}, c_s)}{1 + Y + (Y^2/2)} \]

where

\[ Y = -(C_v)(P-P_{\text{ref}}) \]

**Example**

With NTPVT=2, NPPVT≥4:

```
PVTSALT
--
-- Reference Pressure Reference concentration
```
<table>
<thead>
<tr>
<th>Salt conc</th>
<th>Bw</th>
<th>Cw</th>
<th>Vw</th>
<th>Cv</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>1.01</td>
<td>3.00E-06</td>
<td>0.52</td>
<td>0.0</td>
</tr>
<tr>
<td>2.0</td>
<td>1.00</td>
<td>3.00E-06</td>
<td>0.52</td>
<td>0.0</td>
</tr>
<tr>
<td>5.0</td>
<td>0.98</td>
<td>3.00E-06</td>
<td>0.52</td>
<td>0.0</td>
</tr>
<tr>
<td>20.0</td>
<td>0.95</td>
<td>3.00E-06</td>
<td>0.52</td>
<td>0.0</td>
</tr>
</tbody>
</table>

---

**Reference Pressure**

---

<table>
<thead>
<tr>
<th>Salt conc</th>
<th>Bw</th>
<th>Cw</th>
<th>Vw</th>
<th>Cv</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>1.00</td>
<td>3.00E-06</td>
<td>0.50</td>
<td>0.0</td>
</tr>
<tr>
<td>20.0</td>
<td>0.95</td>
<td>4.00E-06</td>
<td>0.60</td>
<td>0.0</td>
</tr>
</tbody>
</table>

---
PVT properties of dry gas (using Z-factors)

The data comprises NTPVT (see item 2 of keyword TABDIMS in the RUNSPEC section) tables of dry gas PVT functions. Each table consists of two records, each terminated by a slash (/).

**Record 1**

One item of data:

1. The reference temperature for this table.

   **UNITS:** °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

   The temperature is used to convert the Z factor into a formation volume factor.

**Record 2**

A table consisting of 3 columns of data:

1. The gas phase pressure.

   Values should increase monotonically down the column.

   **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2. The corresponding compressibility (Z) factor.

3. The corresponding gas viscosity.

   Values should be level or increasing down the column.

   **UNITS:** cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

   There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (item 4 in keyword TABDIMS).

The Z factor is related to the formation volume factor $B_g$, reference temperature $T_{ref}$ and pressure $P$ by

$$B_g = Z \times \left( \frac{T_{ref} + T_{base}}{T_s + T_{base}} \right) \times \frac{P_s}{P}$$

**Eq. 3.162**

where $P_s$ and $T_s$ denote pressure and temperature at standard conditions, taken as

- $P_s = 1.01325$ barsa (METRIC), 14.7 psi (FIELD), 1.0 atma (LAB)
- $T_s = 15.56 \times ^\circ C$ (METRIC), 60 \times ^\circ F (FIELD), 15.56 °C (LAB)

and $T_{base}$ is the absolute temperature at base measured temperature given by

- $T_{base} = 273.105$ K (METRIC), 459.598 \times ^\circ R (FIELD), 273.105 K (LAB).

See also keyword PVDG.

**Example**

With NTPVT=1 and NPPVT=8:

```
PVT
  -- Temperature
  150 /
```

**Keywords**

PVZG
<table>
<thead>
<tr>
<th>Press</th>
<th>Z</th>
<th>Visc</th>
</tr>
</thead>
<tbody>
<tr>
<td>400.00</td>
<td>1.22</td>
<td>0.01300</td>
</tr>
<tr>
<td>1200.00</td>
<td>1.30</td>
<td>0.01400</td>
</tr>
<tr>
<td>2000.00</td>
<td>1.34</td>
<td>0.01500</td>
</tr>
<tr>
<td>2800.00</td>
<td>1.50</td>
<td>0.01600</td>
</tr>
<tr>
<td>3600.00</td>
<td>1.55</td>
<td>0.01700</td>
</tr>
<tr>
<td>4000.00</td>
<td>1.70</td>
<td>0.01750</td>
</tr>
<tr>
<td>4800.00</td>
<td>1.82</td>
<td>0.01850</td>
</tr>
<tr>
<td>5200.00</td>
<td>1.91</td>
<td>0.01900</td>
</tr>
</tbody>
</table>

/
This section contains an alphabetic listing of keywords starting with the letter Q.
Places wells in the sequential drilling queue

The keyword should be followed by a list of up to 50 well names, terminated with a slash (/).

The wells are placed in the sequential drilling queue, to be opened in sequence when they are needed to maintain a group rate target under group control by guide rate (GCONPROD, GCONINJE, GCONSALE). These wells may also be opened in sequence from the drilling queue in ECLIPSE 100 should they be needed to maintain a group’s production potential (GDRILPOT). If the drilling queue already contains some wells, the new list of wells is added to the end of the queue. Note that there is only one drilling queue - each time QDRILL is entered it appends its wells to the end of this queue. Thus, if there are more than 50 wells to go into the queue, the remainder can be added to the queue by using the keyword a second time. Wells can be removed from the drilling queue, if required, by declaring them OPEN with keyword WCONPROD, WCONINJE or WELOPEN.

If a group (including the field) cannot meet its production target, the first production well in the queue that is subordinate to that group, and not subordinate to any other group under the same mode of production control, is opened automatically. If a group cannot meet its injection target, the first injection well in the queue that is subordinate to that group, and not subordinate to any other group under injection control for the same phase, is opened automatically.

Well opening may be delayed by constraints on the rate of drilling new wells (see keyword WDRILTIM), the availability of drilling rigs (keyword GRUPRIG) and, in ECLIPSE 100, on the maximum number of open wells in any group (item 9 of keyword GECON). If a well cannot currently be drilled because of any of these constraints, the subsequent wells in the drilling queue will be examined in turn to see if there is another suitable well that can be drilled instead.

An alternative to the sequential drilling queue is the prioritized drilling queue, from which wells are opened in decreasing order of their drilling priority (see keyword DRILPRI). Wells are placed in the prioritized drilling queue using the keyword WDRILPRI. Only one type of drilling queue may exist at any given time in the run; the two types of queue cannot operate together.

The drilling queue facility should not be confused with the automatic opening option (AUTO in item 2 of keyword WCONPROD and item 3 of keyword WCONINJE for example). They are two completely separate facilities. Wells in the drilling queue are opened only when needed to maintain group production and injection targets. Wells on AUTO will be opened as soon as allowed by constraints on drilling rate and number of open wells per group, irrespective of group flow rates.

**Note:** The wells must have been fully specified as shut or stopped producers or injectors under independent or group control.

**Note:** The drilling queue is not used for opening wells belonging to groups under prioritization group control (keyword GCONPRI), as the prioritization algorithm itself handles the opening and closing of these wells.

See also keywords WDRILRES, WELSOMIN, PRORDER and WDRILPRI.

**Example**

```
QDRILL
PROD4 PROD5 PROD6
PROD7
```
QHRATING  Relationship of massflow to depth

This keyword is used to define the downstream stream rating table boundary condition of a river.

The keyword is followed by NRATTA tables with a maximum of MXRATE entries in each. These numbers are set with the RIVRDIMS keyword. Each table is terminated with a slash (/).

Each table consists of 2 columns of data:

1. Massflow
   - The first entry in a table must be 0.0
   - UNITS: m$^3$/day (METRIC), ft$^3$/day (FIELD), cc/hr (LAB)

2. Depth of the river at the massflow given in item 1
   - UNITS: m (METRIC), ft (FIELD), cm (LAB)

There is a minimum of two rows of data per table. End the table with a slash (/). After the first tables subsequent tables may be defaulted to the previous table by just entering a slash.

**Example**

```
QHRATING
0.0 0.0
100 1
200 3
/
0.0 0.0
100 1
200 3
/
```
QMOBIL Controls the mobile end-point correction in an LGR

QMOBIL may be used in LGR runs to switch the mobile end-point correction facility on or off within individual local grid refinements. It should be placed between the keyword that sets up the LGR (CARFIN or RADFIN) and the corresponding ENDFIN keyword.

The keyword operates in an analogous manner to the 'MOBILE' option in the EQLOPTS keyword in the RUNSPEC section, which turns on the mobile end-point correction facility in the global grid. By default, if the 'MOBILE' option is switched on, then the mobile end-point correction facility will operate in all local grids as well as in the global grid. Conversely, if the option is not enabled in RUNSPEC, the facility is turned off in the global grid and by default in all local grids. The QMOBIL keyword may be used to override the default setting in individual local grids.

If the mobile end-point correction facility is turned on in any local grid, but is not turned on in the global grid, then the endpoint scaling option must be enabled with the keyword ENDScale in the RUNSPEC section.

The QMOBIL keyword is followed by a record containing one item of data, terminated with a slash, which relates to the local grid system introduced in the preceding CARFIN or RADFIN keyword:

1. Mobile end-point correction flag
   
   YES Mobile end-point correction is required in the local grid
   
   NO Mobile end-point correction is not required in the local grid.
   
   DEFAULT:
   
   YES If the mobile fluid correction is activated for the global grid using EQLOPTS in the RUNSPEC section.
   
   NO Otherwise

Example

If the mobile fluid correction is activated using EQLOPTS in RUNSPEC, and you want to turn the facility off in the local grid NOQMOB.

```
CARFIN
NOQMOB 1 2 1 2 4 4 /
QMOBIL
NO /
ENDFIN
```
This section contains an alphabetic listing of keywords starting with the letter R.
**RADFIN4**

**Specifies a four-column radial local grid refinement**

RADFIN4 sets up a single radial local grid refinement inside a given box of four columns of global grid cells. The box is identified by its global grid coordinates $I_1 - I_2$, $J_1 - J_2$, $K_1 - K_2$. The dimensions of the refined grid within this box of four columns are specified as $NR$, $NTHETA$, $NZ$.

Local grid refinement is described in "Local grid refinement and coarsening" in the *ECLIPSE Technical Description*.

The keyword is followed by the following items of data, terminated with a slash.

1. Name of the local grid refinement - up to 8 characters enclosed in quotes.
2. $I_1$ lower $I$-coordinate of the column in the global grid.
3. $I_2$ upper $I$-coordinate of the box in the global grid.
   
   This must be $I_2 = I_1 + 1$
4. $J_1$ lower $J$-coordinate of the box in the global grid.
5. $J_2$ upper $J$-coordinate of the box in the global grid.
   
   This must be $J_2 = J_1 + 1$
6. $K_1$ minimum $K$-coordinate of the box in the global grid.
8. $NR$ number of radial divisions in the refined grid.
9. $NTHETA$ number of angular segments in the refined grid
   
   Allowed values are $NTHETA = 4$ or $8$.
   
   If a corner point host grid is specified using $COORD / ZCORN$, then only $NTHETA = 8$ is allowed.
   
   By default, the circle is completed. (Use $COORDSYS$ if you wish to override this.)
10. $NZ$ Number of vertical divisions in the refined grid.
11. $NWMAX$ Maximum number of wells this local refined grid will contain.
    
    DEFAULT: 1 (which is the normal value in a radial refinement).

The actual number of wells in any local grid, or amalgamation of local grids, must not exceed $NWGMAX$, in item 4 of the $WELLDIMS$ keyword in the $RUNSPEC$ section.

Any other $GRID$ section keywords (for example $INRAD$ and $COORDSYS$) following this keyword are taken to apply to this local refined grid, until an $ENDFIN$, $CARFIN$, $REFINE$, $RADFIN$ or another $RADFIN4$ keyword is encountered.

See also keywords $RADFIN$, $HRFIN$, $NZFIN$.

**Note:** The inner radius of the radial refinement **must** be specified using keyword $INRAD$ between the $RADFIN4$ and $ENDFIN$ keywords.
The outer radius may be specified using keyword `OUTRAD` but this is not compulsory. If the outer radius is not specified using `OUTRAD`, it defaults to `RMAX/3` where `RMAX` is the maximum radius of a circle contained within the four cells cell containing the radial refinement.

### Example

```
RADFIN4
-- NAME   I1 I2  J1 J2  K1  K2  NR  NTHETA  NZ   NWMAX
'SOUTH'   7  8   8  9   3   5   6   8      18  /
INRAD
0.2   /
ENDFIN
```
**RADFIN**

**Specifies a single-column radial local grid refinement**

The **RADFIN** keyword sets up a single radial local grid refinement inside a given column of global grid cells. The column is identified by its global grid coordinates $I$, $J$, $K_1$, $K_2$. The dimensions of the refined grid within this column are specified as $NR$, $N\theta$, $NZ$. Local grid refinement is described in "Local grid refinement and coarsening" in the **ECLIPSE Technical Description**.

This keyword contains the following items of data, terminated with a slash (/):

1. Name of the local grid refinement - up to 8 characters enclosed in quotes.
2. $I$ I-coordinate of the column in the global grid.
5. $K_2$ Maximum K-coordinate of the column in the global grid.
6. $NR$ Number of radial divisions in the refined grid.
7. $N\theta$ Number of angular segments in the refined grid.
   
   Allowed values:
   
   - 1 - For a 2-D radial grid
   - 4 - For a 3-D radial grid.
   
   If a corner point host grid is specified using **COORD/ZCORN**, then only $N\theta = 4$ is allowed.
8. $NZ$ Number of vertical divisions in the refined grid.
9. $NW\text{MAX}$ Maximum number of wells this local refined grid contains (not used in **ECLIPSE 300**)
   
   **DEFAULT**: 1 (Which is the normal value in a radial refinement.)
10. **ECLIPSE 300 only** Inner radius of refinement
    
    **ECLIPSE 300 only**
    
    *UNITS*: m (**METRIC**), ft (**FIELD**), cm (**LAB**), m (**PVT-M**)
    
    **DEFAULT**: 0.5 ft = 0.1524 m
11. **ECLIPSE 300 only** Outer radius of refinement
    
    **UNITS**: m (**METRIC**), ft (**FIELD**), cm (**LAB**), m (**PVT-M**)
    
    **DEFAULT**: $1/3$ of largest circle that can be inscribed in the column of cells
12. **ECLIPSE 300 only** Minimum $AR$ value allowed in the refinement
    
    **UNITS**: m (**METRIC**), ft (**FIELD**), cm (**LAB**), m (**PVT-M**)
    
    **DEFAULT**: 5 ft = 1.524 m
13. Name of parent LGR (for nested refinements in **ECLIPSE 300 only**) - up to 8 characters. This item may be set to either a null string or to ‘GLOBAL’ to indicate that the parent grid is global (that is, this is not a nested refinement).
The sides and central axis of the radial refinement are assumed vertical as described in "Radial refinements" in the ECLIPSE Technical Description. Departures from this are allowed, but may generate warnings. See "Current restrictions" in the ECLIPSE Technical Description.

**ECLIPSE 300 only**
Nested refinements may be used in ECLIPSE 300. If the parent grid is already an LGR then the name of the parent LGR should be specified. The range of I-, J- and K- indices should then refer to the parent grid. If the parent grid is the global grid, then the range of I-, J- and K- indices refers to the global grid.

**ECLIPSE 100 only**
Nested refinements cannot be used in ECLIPSE 100. This item should be left defaulted or may be set to either a null string or ‘GLOBAL’

DEFAULT: ‘GLOBAL’

**ECLIPSE 100**
The actual number of wells in any local grid, or amalgamation of local grids, must not exceed NWGMAX, in item 4 of the WELLDIMS keyword in the RUNSPEC section.

**ECLIPSE 100**
items 10-13 are unused in ECLIPSE 100 and are ignored. The functionality can however be duplicated using the INRAD, OUTRAD and HRFIN keywords.

Any other GRID section keywords (for example, INRAD and PORO) following this keyword are taken to apply to this local refined grid, until an ENDFIN, CARFIN, REFINE, RADFIN4 or another RADFIN keyword is encountered.

See also keywords RADFIN4, HRFIN, NZFIN.

**Notes**

**ECLIPSE 100**
• The inner radius of the radial refinement must be specified using keyword INRAD between the RADFIN and ENDFIN keywords.

The outer radius may be specified using keyword OUTRAD but this is not compulsory. If the outer radius is not specified using OUTRAD, it will default to RMAX/3 where RMAX is the maximum radius of a circle contained within the cell containing the radial refinement.

• When the grid is a dual or (in ECLIPSE 300) a multi-porosity grid, then the box of the parent grid is specified in the matrix only. The number of refined cells in the Z-direction is specified in the same way as for the DIMENS keyword in RUNSPEC - namely that NZ (item 8) gives the total number of Z-direction cells in all porosities, so that NZ must be a multiple of the number of porosities. An error will be posted if this is not the case. For example, for a dual porosity case, NZ should be a multiple of 2 and NZ/2 should give the number of Z-direction refined cells in the matrix.

**ECLIPSE 300**
• Although the inner radius can be specified using item 10 of the RADFIN keyword, the inner radius value can also be overwritten using the INRAD keyword between the RADFIN / ENDFIN bracket.

**Example**

```
RADFIN       -- NAME     I  J  K1 K2 NR NTHETA  NZ  NWMAX
     \ --- 'SOUTH' 7  8  3  5  6  4  18 / /
     \ INRAD   0.2 / ENDFIN
```

Keywords

1777
Completing the circle

By default, non-neighbor connections are generated to complete the circle for each radial refinement. These connections act between the last cell in the theta-direction, and the first cell, for each layer and ring of cells, in the radial-direction.

By default, the circle is completed for NTETA = 4 refinements.

**ECLIPSE 300 only**

If the circle is not required to be completed, then the keyword NOCIRCLE should be used within the RADFIN / ENDFIN bracket.

**ECLIPSE 100 only**

If the circle is not required to be completed, use the COORDSYS keyword between the RADFIN/ENDFIN bracket. Refer to these keywords for more information.
**RADIAL**

Indicates that the run uses radial geometry

This keyword specifies that radial geometry is to be used. Cell coordinates are specified in terms of R-theta-Z values. Cell dimensions, permeabilities and transmissibilities may then be specified using radial alias keywords, and are interpreted as being in the radial geometry coordinate system.

One additional keyword is required in the **GRID** section for the radial case is **INRAD**, which is used to specify the inner radius of the grid.

For ECLIPSE 100, if the radial grid completes a full circle, the **COORDSYS** keyword in the **GRID** section must be used to tie the two edges of the grid together with non-neighbor connections.

The **RADIAL** keyword has no associated data.
RAINFALL Specifies a monthly varying flux rate for a constant flux aquifer

The keyword should be followed by the following data items and be terminated with a slash.

Note: No entry may be defaulted.

Negative 'influxes' are allowed.

Timesteps are chosen to honor the monthly intervals. This may mean that the minimum timestep given in TUNING is ignored.

1. Aquifer identification number
   This should be a number between 1 and the maximum number of aquifers, set in the keyword AQUIMS and must refer to an AQUFLUX type aquifer.

2. January aquifer flux
   This is the water inflow rate per unit area of connected cell face for the month of January.
   UNITS: sm³/day/m² (METRIC), stb/day/ft² (FIELD), scc/hr/cm² (LAB)

3. February aquifer flux
   This is the water inflow rate per unit area of connected cell face for the month of February.
   UNITS: sm³/day/m² (METRIC), stb/day/ft² (FIELD), scc/hr/cm² (LAB)

4. March aquifer flux
   This is the water inflow rate per unit area of connected cell face for the month of March.
   UNITS: sm³/day/m² (METRIC), stb/day/ft² (FIELD), scc/hr/cm² (LAB)

5. April aquifer flux
   This is the water inflow rate per unit area of connected cell face for the month of April.
   UNITS: sm³/day/m² (METRIC), stb/day/ft² (FIELD), scc/hr/cm² (LAB)

6. May aquifer flux
   This is the water inflow rate per unit area of connected cell face for the month of May.
   UNITS: sm³/day/m² (METRIC), stb/day/ft² (FIELD), scc/hr/cm² (LAB)

7. June aquifer flux
   This is the water inflow rate per unit area of connected cell face for the month of June.
   UNITS: sm³/day/m² (METRIC), stb/day/ft² (FIELD), scc/hr/cm² (LAB)

8. July aquifer flux
   This is the water inflow rate per unit area of connected cell face for the month of July.
   UNITS: sm³/day/m² (METRIC), stb/day/ft² (FIELD), scc/hr/cm² (LAB)
9. August aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of August.
UNITS: $\text{sm}^3/\text{day/m}^2$ (METRIC), $\text{stb/day/ft}^2$ (FIELD), $\text{scc/hr/cm}^2$ (LAB)

10. September aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of September.
UNITS: $\text{sm}^3/\text{day/m}^2$ (METRIC), $\text{stb/day/ft}^2$ (FIELD), $\text{scc/hr/cm}^2$ (LAB)

11. October aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of October.
UNITS: $\text{sm}^3/\text{day/m}^2$ (METRIC), $\text{stb/day/ft}^2$ (FIELD), $\text{scc/hr/cm}^2$ (LAB)

12. November aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of November.
UNITS: $\text{sm}^3/\text{day/m}^2$ (METRIC), $\text{stb/day/ft}^2$ (FIELD), $\text{scc/hr/cm}^2$ (LAB)

13. December aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of December.
UNITS: $\text{sm}^3/\text{day/m}^2$ (METRIC), $\text{stb/day/ft}^2$ (FIELD), $\text{scc/hr/cm}^2$ (LAB)

The water flow rate into a grid block from a constant flux aquifer is given by:

$$Q_w = F_w \cdot A \cdot M$$  \hspace{1cm} \text{Eq. 3.163}$$

- $F_w$ is the flux, entered in item 2
- $A$ is the area of the connected cell face
- $M$ is the aquifer influx multiplier (item 10 of the AQUANCON keyword or item 11 of the AQANCONL keyword), which defaults to 1.0.

where

The area is calculated directly from the connected cell geometry, and is not affected by item 9 of the AQUANCON keyword or item 10 of the AQANCONL keyword.

The aquifer flux can be modified at any time during the simulation by re-entering the RAINFALL keyword in the SCHEDULE section. The keyword replaces any flux entered in the AQUFLUX keyword.

**Example**

```
RAINFALL
  1
  0.1
  0.11
  0.2
  0.15
  0.07
  0.03
  0.02
  0.08
  0.09
```
RBEDCONT Tables of river bed/grid-block contact area data

This keyword is used to define the contact area of a river with a grid-block as a function of the depth.

The keyword is followed by MXTBGR tables with a maximum of MXDPTB entries in each. These numbers are set with the RIVRDIMS keyword. Each table is terminated with a slash (/).

Each table consists of 2 columns of data:

1. Depth
   The values should increase monotonically down the table.
   The first entry should be 0.0
   UNITS: m (METRIC), ft (FIELD), cm (LAB)

2. Contact area at this depth
   The values should increase monotonically down the table.
   UNITS: m² (METRIC), ft² (FIELD), cm² (LAB)

There is a minimum of two rows of data per table. End the table with a slash (/). After the first tables subsequent tables may be defaulted to the previous table by just entering a slash.

Example

To define a single table giving a constant contact area:

```
RBEDCONT
0.0 100.0
10.0 100.0/
```
**RCMASTS**  Sets minimum length for timesteps restricted by flow change

This keyword may be used in a Reservoir Coupling master run to set a minimum length for timesteps restricted by a group’s limiting flow rate fractional change (see item 4 in keyword **GRUPMAST**). If any groups have a limiting flow rate fractional change set, it is advisable to set a minimum timestep length that this restriction can impose, as the fractional flow changes can be large when the flows are small and wells are being opened or closed in the slave reservoirs.

The keyword is followed by a record containing a single number, terminated with a slash (/). The number represents the minimum timestep length that can be imposed by a group’s limiting flow rate fractional change.

**UNITS:** day (**METRIC**), day (**FIELD**), hr (**LAB**), day (**PVT-M**)

**DEFAULT:** TSMINZ in keyword **TUNING**

**Example**

```plaintext
RCMASTS 10.0 /
```
The rate \( R \) of a chemical reaction depends on a rate constant (\( \text{REACRATE} \)), porosity (\( \text{REACPORD} \)), component concentrations (\( \text{REACCORD} \)), component phases (\( \text{REACPHA} \)), and temperature \( T \):

\[
R \propto e^{-\frac{E_r}{RT}}
\]

where

\( R \) is the gas constant and

\( E_r \) is the activation molar energy of the reaction.

This keyword is followed by one item for each reaction defined by the \( \text{REACTION} \) keyword, each item representing the activation molar energy for the corresponding reaction. The list of items is terminated with a slash (/).

UNITs: kJ/kg-M (\text{METRIC}), Btu/lb-M (\text{FIELD}), J/gm-M (\text{LAB}), kJ/kg-M (\text{PVT-M})

DEFAULT: 0

See "Chemical reactions" in the ECLIPSE Technical Description.

Example

For two reactions with activation energies 19500 kJ/kg-M and 13700 kJ/kg-M respectively:

\[
\text{REACACT} \\
19500 \ 13700 \\
/ 
\]
The rate \( R_r \) of a chemical reaction depends on a rate constant (\( \text{REACRATE} \)), the temperature (\( \text{REACACT} \)), porosity (\( \text{REACPORD} \)), component phases (\( \text{REACPHA} \)) and the concentration of reactants:

\[
R_r \propto (m_c)^{n_c}
\]

where

\( m_c \) is the concentration of component \( c \) in the reacting phase and

\( n_c \) is the index or order of the component term defined by this keyword.

The keyword is followed by one record for each reaction defined by the \text{REACTION} keyword. Each record contains \( N_{hc} + N_w \) items, one for each hydrocarbon component and one for each water component in the model. Each item represents the order of the component term for the corresponding reaction. Each record is terminated with a slash (/).

DEFAULT: 0 for each item.

For reactants the order must be non-negative. non-zero orders can be entered for components that are not reactants - in which case, when the component concentration \( m_c > 1 \), the component acts as a catalyst (or inhibitor if the order is negative).

If the order for a component is set to zero the reaction rate is independent of the concentration of that component. This is not recommended for reactants, unless keyword \text{EQLDREAC} is also used.

See "Chemical reactions" in the \textit{ECLIPSE Technical Description}.

**Example**

For a run with four components \( \text{C}_{12}\text{H}_{26}, \text{C}_3\text{H}_8, \text{O}_2, \text{CO}_2, \) and water, and two reactions

\[
\text{C}_{12}\text{H}_{26} + 18.5 \cdot \text{O}_2 \rightarrow 12 \cdot \text{CO}_2 + 13 \cdot \text{H}_2\text{O}
\]

and

\[
\text{C}_3\text{H}_8 + 5 \cdot \text{O}_2 \rightarrow 3 \cdot \text{CO}_2 + 4 \cdot \text{H}_2\text{O}
\]

The following example specifies that the reaction rate of the first reaction depends on the concentration of the first and third components, while the reaction rate of the second reaction depends on the concentration of the second and third components:

```
REACCORD
---HEAVY  LIGHT  O2    CO2  H2O
1      0      1     0    0  /
0      1      1     0    0  /
```

Keywords

1786
**Reaction rate interpolation criteria**

This keyword can be used in Thermal simulations to improve stability when modeling combustion processes.

When calculating the chemical reaction rate, certain terms can be interpolated between their values at the current timestep and the previous timestep, for example, temperature:

\[ T = \theta \cdot T_n + (1-\theta) \cdot T_{n-1}, \quad 0 < \theta \leq 1 \]

where \( \theta \) is an interpolation parameter.

The keyword is followed by the following items of data, terminated with a slash.

1. Interpolation parameter for temperature used in the reaction rate Arrhenius term
   
   **DEFAULT**: 0.5

2. Interpolation parameter for oil saturation used in the reaction rate
   
   **DEFAULT**: 1 (that is, fully implicit)

See "Chemical reactions" in the *ECLIPSE Technical Description*.

**Example**

```
REACCRIT
  1  0.5  /
```
**REACENTH Reaction molar enthalpy**

This keyword can only be used with the Thermal option. It defines the molar enthalpy of a chemical reaction.

The keyword is followed by one item for each reaction defined by the `REACTION` keyword, each item representing the molar enthalpy for the corresponding reaction. The list of items is terminated with a slash (/).

**UNITS:** kJ/kg-M (METRIC), Btu/lb-M (FIELD), J/gm-M (LAB), kJ/kg-M (PVT-M)

**DEFAULT:** 0

See "Chemical reactions" in the *ECLIPSE Technical Description*.

**Example**

For two reactions:

```
REACENTH
  16000 16000 /
```
REACHES

Defines the reaches within a river

This keyword is used to define the reach structure of a river. The river must previously have been introduced with the RIVERSYS keyword.

The keyword can only be used on one river at a time, so it must be specified more than once if there is more than one river.

The keyword is followed by a number of data records to describe the reach properties and geometry of the (possibly multibranched) river. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

The data items in each record are:

Record 1

The first record identifies the river, sets some general flags, and sets the properties to be associated with the river mouth. This reach is labeled reach number 1 and is reserved for internal use by ECLIPSE.

1. Name of the river
2. x-coordinate of the river mouth. (Downstream end of river)
   UNITS: m (METRIC), ft (FIELD), cm (LAB)
3. y-coordinate of the river mouth. (Downstream end of river)
   UNITS: m (METRIC), ft (FIELD), cm (LAB)
4. z-coordinate of the river mouth. (Downstream end of river)
   UNITS: m (METRIC), ft (FIELD), cm (LAB)
5. Length down the river to the river mouth
   This item can be used if the river being modeled within the simulator is not a complete river and it is wished to offset the distances from the true river mouth to the position defined in items 2-4.
   UNITS: m (METRIC), ft (FIELD), cm (LAB)
   DEFAULT: 0.0
6. Type of length and depth information entered in the subsequent records of this keyword
   INC  The length and depth values entered in subsequent records refer to the incremental changes of these quantities along each reach.
   ABS  The length and depth values entered in subsequent records refer to the absolute values of the tubing length and depth at the reach nodes.
   DEFAULT: There is no default; this item must be set.

End the data record with a slash (/).

Each subsequent record

The subsequent records define the properties of all the river’s reaches except for the top reach. The reaches must be numbered between 2 and MXREAC (entered in the RUNSPEC section keyword RIVRDIMS). Reaches may be entered in any order, and neighboring reaches need not be numbered consecutively;
ECLIPSE reorders them internally. A contiguous set of reaches having the same properties can be entered together in a single record as a range. The reaches within a range must have consecutively increasing numbers.

1. Reach number at the start of the range (nearest the river mouth)
   An integer between 2 and MXREAC.

2. Reach number at the far end of the range
   An integer between 2 and MXREAC, not less than item 1.
   The remaining items in this record set the properties of the reaches in this range. To enter properties for a single reach, set items 1 and 2 equal.

3. Number of the reach to which the reach at the start of the range (item 1) is joined, that is its neighbor in the direction of the riverhead. This is its **outlet reach**; the convention being that production flows are positive.

4. The branch name

5. If item 6 of Record 1 is **INC**:
   The length of each reach in the range, that is the distance between its nodal point and that of its neighbor in the direction of the river mouth.
   Or, if item 6 of Record 1 is **ABS**:
   The length down the river (from the river mouth) at the nodal point of the **last** reach in the range. ECLIPSE divides the length between this point and the node of the range’s outlet reach (item 3) into n equal lengths, where n is the number of reaches in this range.
   **UNITS:** m (**METRIC**), ft (**FIELD**), cm (**LAB**)

6. If item 6 of Record 1 is **INC**:
   Depth change along each reach in the range, that is the depth of its nodal point minus the depth of its outlet reach’s nodal point.
   Or, if item 6 of Record 1 is **ABS**:
   The depth of the nodal point of the **last** reach in the range. ECLIPSE calculates the depths of any other reaches in this range by interpolating between this depth and the depth of the range’s outlet reach (item 4), linearly according to their length down the tubing.
   **UNITS:** m (**METRIC**), ft (**FIELD**), cm (**LAB**)

7. Channel profile table number
   **DEFAULT:** 1

8. Manning’s roughness coefficient

9. If item 6 of Record 1 is **INC**:
   The length of each reach in the range projected onto the X-axis, that is the X coordinate of its nodal point minus that of its neighbor in the direction of the river head.
   Or, if item 6 of Record 1 is **ABS**:
   X coordinate of the nodal point of the last reach in the range, relative to the origin of the grid. ECLIPSE calculates the X coordinates of any other reaches in this range by interpolating between this
value and the X coordinate of the range’s outlet reach (item 4), linearly according to their length down the tubing.

The X and Y coordinates of the reach nodes are stored on the restart or graphics files to enable graphics applications to produce plots of the river path. The data is not used for any other purpose and may be left defaulted if path plots are not required.

**UNITS:** m **(METRIC)**, ft **(FIELD)**, cm **(LAB)**

**DEFAULT:** 0.0

10. If item 6 of Record 1 is **INC**:

The length of each reach in the range projected onto the Y-axis, that is the Y coordinate of its nodal point minus that of its neighbor in the direction of the river head.

Or, if item 6 of Record 1 is **ABS**:

Y coordinate of the nodal point of the last reach in the range, relative to the origin of the grid. ECLIPSE calculates the Y coordinates of any other reaches in this range by interpolating between this value and the Y coordinate of the range’s outlet reach (item 4), linearly according to their length down the reach.

**UNITS:** m **(METRIC)**, ft **(FIELD)**, cm **(LAB)**

**DEFAULT:** 0.0

11. If item 6 of Record 1 is **INC**:

The number of refined REACHES within this reach.

**DEFAULT:** 1

12. If item 6 of Record 1 is **INC**:

Item 6 for depth change of next REACH listed downstream. This item is only required with river refinement. This is item 6 of the next record.

**DEFAULT:** No Default.

End the data record with a slash (/).

End the set of data records with a slash.

**Example**

To define a single channel that has two named branches; one having the same name as the river:

```
REACHES
'THUMB' 0.0 0.0 0.0 0.0 inc /
2 8 1 'THUMB' 200 5 1 0.01 2* /
9 13 8 'EYE' 200 5 1 0.02 2* /
```
Pressure and temperature limits for calculating chemical reaction rates

The REACLIMS keyword specifies the pressure and temperature limits used when calculating chemical reaction rates. Block pressures and temperatures outside the range specified are modified to the minimum or maximum values accordingly prior to calculating the reaction rates, see "Chemical reactions" in the ECLIPSE Technical Description.

The keyword is followed by one record for each reaction defined by the REACTION keyword. Each record should contain the following items of data and be terminated with a slash (/).

1. Minimum pressure
   
   UNITS: barsa (METRIC), psia (FIELD), atm (LAB), atma (PVT-M)  
   
   DEFAULT: No minimum

2. Maximum pressure
   
   UNITS: barsa (METRIC), psia (FIELD), atm (LAB), atma (PVT-M)  
   
   DEFAULT: No maximum

3. Minimum temperature
   
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)  
   
   DEFAULT: No minimum

4. Maximum temperature
   
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)  
   
   DEFAULT: No maximum

Example

For a simulation with two reactions, constraining the pressure used to calculate the rate in the first reaction and the temperature in the second:

```
REACLIMS
MIN P  MAX P  MIN T  MAX T
30     70     2*     /  
2*     100    300    /  
```
**REACOPS**

## Options for chemical reaction models

The `REACOPS` keywords specifies options used in the chemical reactions model, see "Chemical reactions" in the *ECLIPSE Technical Description*.

The keyword is followed by three items of data and terminated with a slash (/).

1. **Convergence criteria for the sum of the non-linear residual.** The timestep will only be accepted if the non-linear residual is less than this value. The sum of the non-linear residual can be output in the `SUMMARY` section by including the `NLRESSUM` keyword. This constraint ensures conservation between reacting components. The constraint can be disabled by specifying a negative value.
   
   **DEFAULT:** 1e-4

2. **Convergence criteria for the maximum element of the non-linear residual.** The timestep will only be accepted if the maximum element of the non-linear residual is less than this value. The sum of the non-linear residual can be output in the `SUMMARY` section by including the `NLRESPMAX` keyword. In a similar manner to the first item, this constraint ensures conservation between reacting components. The constraint can be disabled by specifying a negative value.
   
   **DEFAULT:** Disabled

3. **Limit the reaction rates such that only reactants present at the start of the timestep can be consumed within the timestep.** Note with this simplification a component produced by one reaction cannot be consumed by another until the next timestep. Similarly a reactant flowing into a block cannot be consumed until the next timestep.
   
   **YES** or **NO**
   
   **DEFAULT:** NO

4. **Scaling factor to ensure that the reaction rate of zero-order reactants tends to zero as the reactant disappears.** In equilibrium deviation reactions (see the `EQLDREAC` keyword) if a reaction rate is independent of the concentration of a reactant, then the rate is multiplied by a factor $m/(m+s)$ where $m$ is the molar density of the reactant and $s$ is given by this item. Please note that the default value is an experimental value and may be adjusted to ensure that the scaling factor $m/(m+s)$ tends to zero with the concentration of the reactant.
   
   **DEFAULT:** 1.0

---

**Note:** Prior to 2008.1 the convergence criteria on the non-linear residual was not applied. To obtain back compatibility the first item of `REACOPS` should be set negative.

---

**Note:** Simulations containing chemical reactions can be very sensitive. Modifying the items in the `REACOPS` keyword is likely to modify the results of the simulation.

### Example

Tighten the tolerance on the sum of the non-linear residual.

```
REACOPS 1e-5 /
```
The rate $R$ of a chemical reaction depends on a rate constant (REACRATE), the temperature (REACACT), porosity (REACPORD), and component concentrations (REACCORD) and component phases. This keyword allows the phase of each component in the reaction to be specified.

The keyword is followed by one record for each reaction defined by the REACTION keyword. Each record contains $N_c$ items, one for each component, or $N_c + 1$ items if water is included in the model. Each item can be one of 6 values:

- **ALL** if all phases react
- **OIL** if only the oil phase reacts
- **GAS** if only the gas phase reacts
- **GPP** if only the gas phase reacts and the rate depends on the gas partial pressure
- **WAT** if the component is water in the water phase
- **NONE** if the reaction rate is independent of the component.

Each record is terminated with a slash (/).

The default value is **ALL**. Phases are ignored for components with zero order (REACCORD).

If a component can react in two phases, and the reaction rate depends on the phase, then two reactions can be defined with the same stoichiometry, STOREAC and STOPROD, but with different rates REACRATE.

See "Chemical reactions" in the ECLIPSE Technical Description.

**Example**

For a run with four components $C_{12}H_{26}$, $C_3H_8$, $O_2$, $CO_2$ and water, and two reactions with the same stoichiometry:

$$C_3H_8 + 5 \cdot O_2 \rightarrow 3 \cdot CO_2 + 4 \cdot H_2O$$

$$C_3H_8 + 5 \cdot O_2 \rightarrow 3 \cdot CO_2 + 4 \cdot H_2O,$$

that proceed at different rates depending on whether $C_3H_8$ is in the oil or gas phase, the keyword is used to distinguish the two reactions:

<table>
<thead>
<tr>
<th>REACPHA</th>
<th>--HEAVY</th>
<th>LIGHT</th>
<th>O2</th>
<th>CO2</th>
<th>H2O</th>
</tr>
</thead>
<tbody>
<tr>
<td>1*</td>
<td>OIL</td>
<td>GAS</td>
<td>1*</td>
<td>1*</td>
<td>/</td>
</tr>
<tr>
<td>1*</td>
<td>GAS</td>
<td>GAS</td>
<td>1*</td>
<td>1*</td>
<td>/</td>
</tr>
</tbody>
</table>

**Note:** In this example the phase of some components have been defaulted (to **ALL**) even though they are not reactants. This is because the component phase value is not used when the reaction rate is independent of a component.
REACPORD  Order of porosity term in chemical reaction rates

The rate $R_r$ of a chemical reaction depends on a rate constant (REACRATE), the temperature (REACACT), component concentrations (REACCORD), component phases (REACPHA) and the porosity $\theta$:

$$R_r \propto \theta^{n_p}$$

where $n_p$ is the index or order of the porosity term defined by this keyword.

The keyword is followed by one item for each reaction defined by the REACTION keyword, each item representing the order of the porosity term for the corresponding reaction. The list of items is terminated with a slash (/).

The default value depends on the component orders entered with the REACCORD keyword. It is chosen so that the reaction rate depends on the component concentrations per unit cell volume.

This keyword can be used to make a reaction rate independent of porosity by setting the order to zero. In this case the rate depends on the component concentration per unit pore volume.

See "Chemical reactions" in the ECLIPSE Technical Description.

Example

For a run with four reactions that are independent of porosity:

```
REACPORD
  --1  2  3  4
  0  0  0  0 /
```
**REACRATE**  

**Reaction rate constant**

The rate $R_r$ of a chemical reaction depends on the temperature (`REACACT`), porosity (`REACPORD`), component concentrations (`REACCORD`) and phases (`REACPHA`) and a constant term $A_r$:

$$R_r \propto A_r$$

defined by this keyword.

The keyword is followed by one item for each reaction defined by the `REACTION` keyword, each item representing the rate constant for the corresponding reaction. The list of items is terminated with a slash (/).

**DEFAULT:** 0

The units of $A_r$ are not fixed, but are chosen so that the units of $R_r$ are correct. Therefore the units of $A_r$ depend on values entered for the component concentration orders (`REACCORD`). See "Reaction rate units" in the ECLIPSE Technical Description.

**Example**

For two reactions with rates $1 \times 10^6$ and $0.3 \times 10^6$ respectively:

```
REACRATE
  1.0E6  3.0E5 /
```

---

**Keywords**

1796
**REACSMAX  Maximum solid saturation in chemical reaction rates**

In chemical reactions involving solid components, the rate $R_r$ of a chemical reaction depends on the solid saturation $S_s$:

\[ R_r \propto \left( 1 - \frac{S_s}{S_{smax}} \right)^{n_s} \]

where $\hat{S}_{smax}$ is the limiting solid saturation, defined by this keyword and $n_s$ is the order of the solid term defined by the REACSORD keyword.

If the solid saturation in a cell exceeds the value $\hat{S}_{smax}$, the reaction rate in the cell is set to zero. Solid saturations can exceed this limit, because of solid expansion, caused by changes in pressure or temperature in the cell.

The keyword is followed by one item for each reaction defined by the REACTION keyword, each item representing the limiting solid saturation term $0.1 < \hat{S}_{smax} \leq 1$ for the corresponding reaction. The default value is $\hat{S}_{smax} = 1$. The list of items is terminated with a slash (/).

See "Reactions involving solid components" in the ECLIPSE Technical Description.

**Example**

For a run with four reactions:

```
REACSMAX
   -1   2   3   4
   1   1   1   0.95 /
```
### REACSORD  Order of solid term in chemical reaction rates

In chemical reactions involving solid components the rate $R_r$ of a chemical reaction depends on the solid saturation $S^s$:

$$ R_r \propto \left(1 - \frac{\hat{S}^s}{\hat{S}_{smax}}\right)^{n_s} $$

where $\hat{S}_{smax}$ is the limiting solid saturation, defined by the REACSMAX keyword and $n_s$ is the index or order of the solid term defined by this keyword.

The keyword is followed by one item for each reaction defined by the REACTION keyword, each item representing the order of the solid term for the corresponding reaction. The list of items is terminated with a slash (/).

The default values for $n_s$ are explained in "Reactions involving solid components" in the ECLIPSE Technical Description.

### Example

For a run with four reactions:

```plaintext
REACSORD
--1  2  3  4
  1* 1* 1* 2 /
```
**REACTION**

**Number of chemical reactions**

The `REACTION` keyword defines the number of chemical reactions that will be simulated. It can only be used in fully implicit runs.

The keyword should be followed by some or all of the following items of data:

1. The number of reactions.
2. The total number of equilibrium deviation terms (see `EQLDREAC`).
3. The total number of equilibrium k-values (see `EQLDKVCR`).
4. The total number of tracer dependent rate terms (see `REACTRAC`).
5. The total number of equilibrium tables (see `EQLDTAB`).

See "Chemical reactions" in the *ECLIPSE Technical Description*.

---

**Note:** The CPR linear solver (see `CPR` keyword) has been found to improve performance for non-thermal models with Chemical Reactions. Please consider modifying the default linear solver for such models.

---

**Example**

```
REACTION
 2 /
```
**REACTRAC**  **Tracer dependent reaction rates**

The keyword is followed by a number of records, each specifying a tracer-dependent reaction rate. The number of records is specified by the fourth item in the `REACTION` keyword. Each record contains four items of data and is terminated by a slash (/).

1. **The reaction number** \( r \)
2. **The tracer name**
3. **The tracer phase** \( p \), which can be **ALL**, **OIL**, **GAS** or **WAT**
   - DEFAULT: **ALL**
4. **The reaction order** \( n \)
   - DEFAULT: 1.0

The reaction rate for reaction \( r \) is multiplied by a factor 

\[
(\theta \cdot t \cdot c_c)^n
\]

where

- \( \theta \) is the porosity
- \( t \) is the tracer concentration

and \( c_c \) is the concentration of the tracer carrier component \( c \) in the tracer phase \( p \)

\[
c_c = m_c \quad \text{if the tracer phase is ALL}
\]

\[
c_c = b_o \cdot S_o \cdot x_c \quad \text{if the tracer phase is OIL}
\]

\[
c_c = b_g \cdot S_g \cdot y_c \quad \text{if the tracer phase is GAS}
\]

\[
c_c = b_w \cdot S_w \quad \text{if the carrier is water and the tracer phase is WAT}
\]

The tracer names and carrier components must be defined by the `TRACER` keyword.

See "Chemical reactions" in the *ECLIPSE Technical Description*

**Example**

In this example the reaction rate for reactions 1 will be multiplied by 

\[
c_c = \theta \cdot t \cdot b_w \cdot S_w
\]

where \( t \) is the concentration of tracer TRW. In this example, item 4 of `REACTION` is set to 1:

```
REACTRAC
  -- R TRACER PHASE N
  1 TRW WAT 1 /
```
**READDATA**  Read input data from an external source

The **READDATA** keyword allows ECLIPSE to read files of **SCHEDULE** section data generated by an external source during the course of the simulation. This allows the engineer, either by hand or by using another program, to modify the **SCHEDULE** data in response to the simulation results.

The keyword should be followed by one item of data:

1. **Data input method:**
   
   'FILE' Requests the input to be read from a file. This is the only form of data input.
   
   DEFAULT: 'FILE'

**Input by file**

After the **READDATA** keyword has been read, ECLIPSE will expect to read a file of **SCHEDULE** section data after each Restart file has been output, typically at each report time (see keyword **RPTRST**). The input file behaves exactly like an **INCLUDE** file, and can contain any keyword available in the **SCHEDULE** section. The file is a standard free format input file, and it must be named as follows:

root.i0034 or ROOT.I0034 for report set 34 (for example).

The prefix is the root name of the run, and the suffix is numbered with the report number of the corresponding restart file. If the 3 character suffix system is used, the file names are as follows:

- **root.i01 - root.i99** for report numbers 1 to 99
- **root.j00 - root.j99** for report numbers 100 to 199
- **root.k00 - root.k99** for report numbers 200 to 299

To prevent ECLIPSE reading a file which has only been partly written, a lockout system is employed. ECLIPSE checks for the existence of an additional guard file: **root.ok** or **ROOT.OK**.

This guard file needs to be generated for every report step since its presence signals to ECLIPSE that it can now read the data file. If the guard file is not present in the current working directory, ECLIPSE waits until it is present. When the guard file exists, ECLIPSE reads the expected data file and then deletes the guard file.

**Example**

Since **FILE** is the only form of data input, the above example requests the input to be read from a file, by default:

```
READDATA
/
```
RECOVERY

Recovery plant table

This keyword enables tables to be entered to control recovery plants. The recovery plant option is discussed in “The recovery plant option” in the ECLIPSE Technical Description.

The keyword has two items of data, followed by the table.

1. The lower component index used to define the heavy fraction.
   This defaults to the last component.

2. The upper component index used to define the heavy fraction.
   This defaults to the last component.

3. A table of \(N_c + 1\) columns, where \(N_c\) is the number of components in the run, listing the NGL recovery fractions of each component as a function of the heavy mole fraction.
   The maximum number of rows of data in this item is specified using GPTDIMS item 3.
   Each value of the heavy mole fraction should be followed by \(N_c\) values (where \(N_c\) is the number of components in the run). The recovery fraction is the number of moles of NGL per mole of feed for each component.
   Thus there should be \(N_c + 1\) numbers for each value of the heavy mole fraction.

The defaults for items 1 and 2 imply that the recovery plant table is normally of recovery fractions against the mole fraction of the last component in the feed.

**Example**

```
RECOVERY
  9  9
  --Z9 CO2 N2  C1  C2  C3  C4-6 CZ7 CZ8 CZ9
  0.0 0.0 0.002 0.001 0.45 0.65 0.98 1.0 1.0 1.0 --NGL recov. at Z9=0
  0.05 0.0 0.002 0.001 0.48 0.68 0.99 1.0 1.0 1.0 --NGL recov. at Z9=0.05
```

# REFINES

## Initiates data input for a named local grid

This keyword is used to indicate that subsequent data refers to a named local grid. The keyword is followed by the local grid name, terminated with a slash. Subsequent keywords, from the available sets described below, are taken to refer to the specified local grid, until either an \texttt{ENDFIN} or another \texttt{REFINE} keyword is entered.

Local grid refinement is described in "Local grid refinement and coarsening" in the \textit{ECLIPSE Technical Description}.

The \texttt{REFINE} keyword is available in the \texttt{GRID}, \texttt{EDIT}, \texttt{PROPS}, \texttt{REGIONS}, \texttt{SOLUTION} and \texttt{SCHEDULE} sections.

\text{ECLIPSE 100} imposes the following restrictions on the keywords that can be used with this keyword.

### GRID section

The \texttt{REFINE} keyword can be used in the \texttt{GRID} section to return to a local grid that has previously been introduced with \texttt{CARFIN} or \texttt{RADFIN}, to make further modifications to its grid data or to supply additional data.

In addition, the \texttt{BOX}, \texttt{COPY}, \texttt{EQUALS}, \texttt{MULTIPLY} and \texttt{COPYBOX} keywords can be used to set or modify \texttt{GRID} section keywords in regions of the local grid.

### EDIT section

The following grid arrays may be edited for a local grid:

\begin{verbatim}
PORV  TRANX  TRANR  DIFFX  DIFFR
DEPTH  TRANY  TRANTHT  DIFFY  DIFFHT
       TRANZ  DIFFZ
\end{verbatim}

In addition, the \texttt{BOX}, \texttt{COPY} and \texttt{EQUALS} keywords can be used to set or modify the above keywords in regions of the local grid. If any grid block values are defaulted, the host global grid cell value is used.

### PROPS section

In end point scaling runs (keyword \texttt{ENDSCALE} in the \texttt{RUNSPEC} section) the following keywords and their directional equivalents are available:

\begin{verbatim}
SWL,   SWCR,   SWU
SG,    SGCR,   SGU
SOWCR, SOGCR
KRO,   KROW,  KROG
IKRO,  IKROW, IKROG
KRW,   KRW,   IKRW, IKRWR
\end{verbatim}
KRG,  KRGR,  IKRG,  IKRGR

SWATINIT

In vertical equilibrium runs, with vertical equilibrium in the local grid, the VEFRACV keyword is also available.

In addition, the BOX, COPY, EQUALS and OPERATE keywords can be used to set or modify the above keywords in regions of the local grid. If any grid block values are defaulted, the host global grid cell value is used.

**REGIONS section**

The following keywords are available for local grid data input following the REFINE keyword:

PVTNUM
SATNUM
EQLNUM
KRNUMX  KRNUMY  KRNUMZ  (Directional K_r cases)
KRNUMX-  KRNUMY-  KRNUMZ-
IMBNUM  IMBNUMY  IMBNUMZ  (Hysteresis cases)
IMBNUMX  IMBNUMY-  IMBNUMZ-
IMBNUMX-
ROCKNUM  (Rock crushing cases)
MISCNUM  (Miscible cases)
WH2NUM  WH3NUM  (WAG hysteresis cases)

In addition, the BOX, COPY, ADD and EQUALS keywords can be used to set or modify the above keywords in regions of the local grid. If any grid block values are defaulted, the host global grid cell value is used.

**SOLUTION section**

In the SOLUTION section, the REFINE keyword can be used when the run is initialized by enumeration, to specify the initial conditions in the local grid cells. If the initial conditions for the local grid cells are not set in this way, they default to the initial conditions in their global grid host cells.

The REFINE keyword can be followed by any of the enumeration keywords that specify initial conditions on a cell by cell basis:

PRESSURE
SWAT
SGAS

**Keywords**

ECLIPSE Reference Manual

1804
SSOL
SURF
SFOAM
RS or PBUB
RV or PDEW

ECLIPSE 100 only

The **BOX, EQUALS, ADD, MULTIPLY, COPY** and **OPERATE** keywords are not available in the **SOLUTION** section.

**SCHEDULE section**

The **REFINE** keyword can be used in the **SCHEDULE** section to modify region data during a simulation, though modification of **SATNUM** or **PVNUM** data during a simulation should be carried out with care and is **not** recommended in general.

The keyword may be used with the **BOX, EQUALS, ADD** or **COPY** modifiers.

The following keywords are available for local grid data input following the **REFINE** keyword:

**PVNUM**

**SATNUM**

**KRNUMX  KRNUMY  KRNUMZ** (Directional \(K_r\) cases)

**KRNUMX-  KRNUMY-  KRNUMZ-**

**IMBNUM  IMBNUMY  IMBNUMZ** (Hysteresis cases)

**IMBNUMX  IMBNUMY-  IMBNUMZ-**

**IMBNUMX-**

**ROCKNUM** (Rock crushing cases)

**MISCNUM** (Miscible cases)

**MULTX**

**MULTY**

**MULTZ**

**MULTPV**

**EXCAVATE**
Note: If region numbers are modified for refined cells in the global grid, then this change is not reflected in the associated local grid cells unless the local grid region numbers are also modified using the REFINE/ENDFIN pair.

The block of data initiated by the REFINE keyword is terminated by either an ENDFIN keyword or another REFINE keyword.

Examples

EDIT section example

```plaintext
REFINE
'LGR1'
EQUALS
'PORV' 100.0 1 2 1 2 3 4
'DEPTH' 1000.0 1 4 1 4 4 4
ENDFIN
```

PROPS section example

```plaintext
REFINE
'LGR1'
SWL
300*0.25
EQUALS
'SWL' 0.20 1 10 1 10 2 2
/
COPY
'SWL' 'SWCR'
/
ENDFIN
```

REGIONS section example

```plaintext
REFINE
'EAST'
SATNUM
20*1
20*2
20*1
EQUALS
'SATNUM' 1 2 4 2 5 2 2
/
ENDFIN
```

SOLUTION section example

```plaintext
REFINE
'LGR1'
PRESSURE
16*3900
16*3905
16*3910
16*3915
SWAT
32*0.22
16*0.25
16*0.28
/
ENDFIN
```
SCHEDULE section example

```
REFINE
  'EAST'  /
SATNUM
  20*2
  20*1
  20*2 /
EQUALS
  'SATNUM'  1  2  4  2  4  1  4 /
/
ENDFIN
```
REGDIMS

Regions dimension data

The data consists of some or all of the following items which describe the maximum number of regions associated with miscellaneous keywords in the GRID, EDIT, PROPS, REGIONS or SOLUTION section.

The data must be terminated by a slash (/).

1. The maximum number of fluid-in-place regions (NTFIP) defined with keyword FIPNUM in the regions section. A balance sheet showing all fluid movements between fluid-in-place regions can be generated at each report time. If additional sets of fluid-in-place regions have been defined with keyword FIP, the maximum region number in this keyword must also not exceed this number.

Since the fluid-in-place balance sheet reports region-to-region flows, the storage required is proportional to the square of NTFIP. Very large values of NTFIP should therefore be avoided, as excessive storage may be required and in addition the restart files will be very large.

This quantity may also be defined by item 5 of TABDIMS. If it is set in both places, the greater of the two settings is used. Additionally, if item 1 of DYNRDIMS is defined, the greatest of the three settings is used.

DEFAULT: 1

2. The number of sets of fluid-in-place regions (NMFIPR). The standard set is defined by FIPNUM in the REGIONS section. Any additional sets are defined by FIPxxx keywords in the REGIONS section.

DEFAULT: 1

---

ECLIPSE 300

Note: If dynamic regions are used in addition to additional sets of standard fluid-in-place regions, this value needs to be the actual number of specified sets of fluid-in-place regions, not the maximum number.

---

ECLIPSE 100 only

3. The maximum number of independent reservoir regions (NRFREG). (See "Independent reservoir regions" in the ECLIPSE Technical Description and the keyword ISOLNUM in the GRID section.)

DEFAULT: 0

4. The maximum number of flux regions for the Flux option (NTFREG); or the maximum number of regions used by the FLUXNUM keyword in the GRID section.

This quantity may also be defined by item 11 of TABDIMS; if it is set in both places, the maximum of the two is used.

DEFAULT: 0

---

ECLIPSE 100 only

Note: If this value is defaulted or set to 0 and the 2nd item of GRIDOPTS is also defaulted or set to 0, the simulator behaves as though all grid blocks are in MULTNUM region 1. This has no effect except that specifying multipliers for region 1 will not generate a warning or error, and is a side-effect of the default value for MULTNUM being 1.

---

ECLIPSE 300 only

5. The maximum number of user-defined regions to be used in the enhanced TRACK option.

DEFAULT: 0

Keywords

1808
CAUTION: This number should, if possible, be no larger than the actual number of regions to be used, as any 'surplus' represents, depending on how many components are to be tracked, a considerable overhead in terms of both storage and CPU.

6. Coal Bed Methane option: The number of Coal Bed Methane regions, \texttt{NTCREG}, as used by \texttt{COALNUM} in the \texttt{GRID} section.

   DEFAULT: 1

7. The maximum number of user-defined regions used by the \texttt{OPERNUM} keyword in the \texttt{GRID} and \texttt{REGIONS} sections.

   DEFAULT: 0

8. The maximum number of double-precision work arrays for use with the \texttt{OPERATE} and \texttt{OPERATER} keywords. An integer value between 1 and 9999 should be specified.

   DEFAULT: 0

9. The maximum number of integer work arrays for use with the \texttt{OPERATE} and \texttt{OPERATER} keywords. An integer value between 1 and 999 should be specified.

   DEFAULT: 0

\textit{ECLIPSE 100 only} 10. The number of polymer flooding mixing regions, \texttt{NPLMIX}, as used by \texttt{PLMIXNUM} in the \texttt{GRID} section.

   Default: 1

\textbf{Example}

```
REGDIMS
5 5 0 1 /
```
RESIDNUM VE residual flow region numbers

This keyword is optional and should only be used in runs employing the vertical equilibrium option (keyword VE in the RUNSPEC section). The residual flow region number specifies which set of saturation function tables (input using SGFN, SOF3 and related keywords in the PROPS section) should be used in the Vertical Equilibrium residual flow calculation.

The keyword should be followed by one integer for every grid block in the current input box, specifying the residual flow region to which it belongs. The region number should not be less than 1 or greater than NTSFUN (see keyword TABDIMS). The data must be terminated by a slash (/). Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

If the keyword is used, the usual vertical equilibrium flow (calculated using the SATNUM saturation function tables) is modified by adding a second flow component based on the cell saturation and the RESIDNUM relative permeability table.

If water floods a cell containing oil, then a residual oil saturation will be left below the contact. The residual flow calculation is intended to model the slow movement of this residual oil during the period after the initial water influx. The residual flow eventually reduces the residual oil values from the SATNUM value to that of the RESIDNUM table. The rate at which this reduction occurs depends on the value of the relative permeability between these two saturations. Typically the relative permeability values is zero for phases with no residual flow and small otherwise. A warning message is issued if the oil relative permeability exceeds 5% for any residual flow table.

Any capillary pressure values entered in the RESIDNUM saturation tables will be ignored.

See also the saturation function keywords SGFN, SWFN, SOF2, SOF3, SOF32D, SGOF, SLGOF and SWOF, and "Vertical equilibrium" in the ECLIPSE Technical Description.

Example

With NTSFUN=3 specified in keyword TABDIMS; NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS; and no input BOX set:

| RESIDNUM | 240*3 / |

Keywords

ECLIPSE Reference Manual
RESOPT

Request a reservoir optimization simulation

This keyword activates the Reservoir Optimization license key. It requires no data.

For further information, see the "Optimization" chapter in the ECLIPSE Technical Description.

Note: The reservoir optimization feature is not currently available with the parallel option. A list of options that are not compatible with the reservoir optimization feature is given in the "Screening your model for suitability" in the ECLIPSE Technical Description.
**RESORB**

**Coal Bed Methane re-adsorption factor**

This keyword is used to specify a re-adsorption factor for each component in a Coal Bed Methane run. Using a value other than unity scales the rate of re-adsorption of the gas. A zero value prevents gas re-adsorption.

A coal bed methane model can be divided into one or more regions. The number of regions is \(NTCREG\) and this number is input by \(REGDIMS\) item 6. Coal regions are input with the keyword \(COALNUM\). There is one \(RESORB\) record for each coal region up to \(NTCREG\) records. Each record contains the re-adsorption factor for the total number of components defined by keyword \(NCOMPS\).

1. \(r_1 r_2 r_3 \ldots r_{ncomps}\)

   Values should be in \([0,1]\)

   DEFAULT: 1.0

   Re-adsorption factors for coal region number 1.

2. \(r_1 r_2 r_3 \ldots r_{ncomps}\)

   Re-adsorption factors for coal region number 2.

The number of records should equal the maximum number of coal regions.

**Example**

```
RESORB
   --For N2 C1 CH4
   1.0 0.9 0.9 / -- coalnum=1
```

Keywords

1812
RESTART

Read solution from a restart or a save file

The keyword should be followed by a line containing between 2 and 4 items of data, terminated by a slash (/).

1. The root name of the Restart file or Save file.
2. The number of the report from which the run is to be restarted.

**Note:** When using a restart file, the report number does not reset to zero after each 10000 reports (for example the restart file produced after '.X9999' is '.Y0000' but the report numbers are still 9999 and 10000 respectively). See the ECLIPSE File Formats Reference Manual for more details.

**Note:** If the run which generated the file referred to in item 1 had the RPRST mnemonic BASIC=6, then each timestep in that run will have generated restart or save file information. In this case, the report number that needs to be entered here is the timestep number. This information can be obtained from the print file for that run.

3. If the time-dependent data is stored on the Save file instead of the Restart file, the string SAVE should be entered here. Default this item if the time-dependent data is to be read from the Restart file.

4. A string denoting the formatted/unformatted status of the Save file from which the time-dependent data is to be read.
   - UNFORMATTED The SAVE file is unformatted
   - FORMATTED The SAVE file is formatted

This item is only relevant if the time-dependent data is to be read from the Save file (that is item 3 is SAVE).

**DEFAULT:** UNFORMATTED

See also keyword SAVE in the RUNSPEC section, and keywords LOAD, RPRST, RPTSCHED and SKIPREST.

**Example**

```
RESTART
BASE 11 /
```

causes the current run to restart from report time 11 of a previous run whose root file-name was BASE, reading the solution data from the Restart file.

For more information see "Restarts" in the ECLIPSE Technical Description
**RESVNUM**  
Initiates coordinate data input for a given reservoir

This keyword is optional, and is used to allow COORD data to be read in on a reservoir-by-reservoir basis. This is useful when two or more reservoir models are being combined into a single larger grid.

The keyword is followed by a single integer to specify the number of the next reservoir to be input using the COORD keyword. The reservoir number should be between 1 and NUMRES. NUMRES should be set > 1 by the keyword NUMRES in the RUNSPEC section.

See also the COORD and COORDSYS keywords and "Independent reservoir regions" in the ECLIPSE Technical Description.

**Example**

Where NDIVIX=3, NDIVIY=2 and NDIVIZ=4 are specified in RUNSPEC keyword DIMENS and NUMRES =2:

```
RESVNUM
  1 /
  COORD
    0  0  0  0  0  1000
    1000  0  0  1000  0  1000
    2000  0  0  2000  0  1000
    3000  0  0  3000  0  1000
    0  2000  0  0  2000  1000
    1000  2000  0  1000  2000  1000
    2000  2000  0  2000  2000  1000
    3000  2000  0  3000  2000  1000
    0  4000  0  0  4000  1000
    1000  4000  0  1000  4000  1000
    2000  4000  0  2000  4000  1000
    3000  4000  0  3000  4000  1000
  / RESVNUM
  2 /
  BOX
    1  3  1  1  1  4 /
  COORD
    6000  0  0  6000  0  1000
    7000  0  0  7000  0  1000
    8000  0  0  8000  0  1000
    9000  0  0  9000  0  1000
    6000  2000  0  6000  2000  1000
    7000  2000  0  7000  2000  1000
    8000  2000  0  8000  2000  1000
    9000  2000  0  9000  2000  1000
  / BOX
    1  3  2  2  1  4 /
  COORD
    6000  2000  0  6000  2000  1000
    7000  2000  0  7000  2000  1000
    8000  2000  0  8000  2000  1000
    9000  2000  0  9000  2000  1000
    6000  4000  0  6000  4000  1000
    7000  4000  0  7000  4000  1000
    8000  4000  0  8000  4000  1000
    9000  4000  0  9000  4000  1000
  /```
### Controls on debug output for individual rivers

This keyword is used control debug output for individual rivers.

**Note:** This output is primarily intended as an aid to program development and debugging, and it should not be requested for normal program runs.

The keyword is followed by any number of lines, each containing the following data and each terminated with a slash (/). The last line must be followed by a line containing only a slash.

1. River name.
2. Debug output control.
   - This is a 4-figure integer:
     - The 1\textsuperscript{st} figure controls debug from river flow and property calculations.
     - The 2\textsuperscript{nd} figure controls debug from connection flow calculations.
     - The 3\textsuperscript{rd} figure is currently unused.
     - The 4\textsuperscript{th} figure controls debug from river terms in the solver.
   - A value of 1 or more turns the relevant debug output switch on. Higher integers result in greater quantities of debug output. A value of zero turns the debug output off.

### Example

```
RIVDEBUG
RIVER1 2120 /
RIVER2 1001 /
```

---

**Keywords**

- ECLIPSE 100
- ECLIPSE 300
- SPECIAL
- RUNSPEC
- GRID
- EDIT
- PROPS
- REGIONS
- SOLUTION
- SUMMARY
- SCHEDULE
RIVERSYS  Defines the branch structure and boundary conditions of a river

This keyword is used to define the branch structure of a river and the boundary conditions to be associated with those branches, if any. This keyword has the effect of introducing the river.

The keyword can only be used on one river at a time, so it must be specified more than once if there is more than one river.

The keyword is followed by a number of data records to describe the reach properties and geometry of the (possibly multi-branched) river. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

The data items in each record are:

**Record 1**

1. Name of the river.
2. Equation set to be solved
   - ‘KINEMATI’  Kinematic wave equations
   - ‘DIFFUSIV’  Diffusive wave equations
   - ‘STEADYST’  Steady-state wave equations
   - ‘STVENANT’  Time-dependent St. Venant equations

   DEFAULT: No default.

End the data record with a slash (/).

**Each subsequent record**

The subsequent records define the branch names.

1. Branch number.
   
   An integer between 1 and $\text{MXBRRI}$. Branch 1 is reserved for main river, or assigned to a channel that is the same as main river.

2. Branch name, used for further referencing of branches.

   DEFAULT: If defaulted the branch will be given a name that is the branch number concatenated with ‘BRANCH’. For example, branch number 7 would be given a branch name ‘BRANCH07’. Do not default branch names with branch numbers greater than 99.

   The main channel/river must have a branch number of 1. Lateral branches should be numbered between 2 and $\text{MXBRRI}$. Lateral branches may also have subordinate branches; in this case each branch must have a higher branch number than its parent branch.

   The remaining items in this record set the boundary conditions for this branch, if any, and can be entered in any order.

If the branch has a downstream boundary condition then one of the following can be input:

* DOWNSTREAM DEPTH TABLE=n (see keyword DEPTHTAB)
• DOWNSTREAM RATING TABLE=n,m (see keyword QHRATING, n is the table number to use when \( \frac{dh}{dl} > 0 \) and m is the table number to use when \( \frac{dh}{dl} < 0 \). In this release they must be equal)

• DOWNSTREAM UNIFORM_FLOW (This boundary condition requires no data and assumes uniform flow conditions at outflow.)

**Note:** There must be no spaces around the ‘=’ or the ‘,’ and the ‘_’ is significant.

If the branch has an upstream boundary condition then one of the following may be input:

• UPSTREAM MASSFLOW TABLE=n (see keyword MASSFLOW)

End the data record with a slash (/).

End the set of data records with a slash.

**Examples**

**Example 1**

To define a single channel that has two named branches; one having the same name as the river:

```
RIVERSYS
 'MYRIVER' /
 1 MYRIVER DOWNSTREAM DEPTH TABLE=1 / 
 2 MYTRIB UPSTREAM MASSFLOW TABLE=1 /
 /
```

**Example 2**

To define a single channel that has just the one branch having the same name as the river:

```
RIVERSYS
 'MYRIVER' /
 1 MYRIVER DOWNSTREAM DEPTH TABLE=1 UPSTREAM MASSFLOW TABLE=1 /
 /
```
**River dimension data**

The data consists of some or all of the following items, describing the dimensions of the river data to be used in the run. The data must be terminated by a slash (/).

A Schlumberger Internal license controlling the River facility can be obtained by contacting ECLIPSE support.

1. The maximum number of rivers in the model.
   - DEFAULT: 0
2. The maximum number of reaches per river.
   - This number should always be set one greater than the number of reaches input in the `REACHES` keyword as ECLIPSE always reserves the first reach for internal use.
   - DEFAULT: 1
3. The maximum number of branches in a river (including the main channel).
   - DEFAULT: 1
4. The maximum number of grid blocks to be associated with any one river. (See `COMPRIV`)
   - DEFAULT: 1
5. Number of tables describing channel profile, `MXTBPR`. (See `RIVRXSEC`)
   - DEFAULT: 1
6. Maximum number of depth entries in a channel profile table, or bed contact area table, `MXDPTB`. (See `RIVRXSEC` or `RBEDCONT`)
   - DEFAULT: 2
7. Number of tables describing contact areas between grid blocks and the river bed, `MXTBGR`. (See `RBEDCONT`)
   - DEFAULT: 1
8. Number of depth tables used as boundary conditions, `NMDEPT`. (See `DEPTHTAB`)
   - DEFAULT: 0
9. The maximum number of entries in a downstream boundary condition depth table, `MXDEPT`.
   - DEFAULT: 2
10. The number of massflow (boundary condition) tables, `NMMAST`. (See `MASSFLOW`)
    - DEFAULT: 0
11. The maximum number of entries in a massflow table, `MXMAST`.
    - DEFAULT: 2
12. Number of rating tables, `NRATTA`. (See `QHRATING`)
    - DEFAULT: 0
13. The maximum number of entries in a rating table, `MXRATE`.
DEFAULT: 2

Example

```plaintext
RIVRDIMS
1 40 2 40 10 20 1 1 50 1 50 /
```
RIVRPROP  Modifies properties of river reaches

This keyword may be used to modify properties of individual reaches in a river (see "Rivers" in the ECLIPSE Technical Description) during the simulation, as an alternative to reentering the reach data with keyword REACHES.

The keyword is followed by a number of data records, each containing the following items of data. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

1. River name
   
The river must previously have had its reaches defined with keyword RIVERSYS.

2. Reach number at the start of the range
   
   An integer between two and the maximum number of reaches defined by item two in the RIVRDIMS keyword.

3. Reach number at the end of the range
   
   An integer between two and the maximum number of reaches defined by item two in the RIVRDIMS keyword.
   
   The following reach properties in this record apply to all the reaches in the range. To enter properties for a single reach, set items two and three equal.

4. Manning roughness coefficient
   
   UNITS: dimensionless
   
   DEFAULT: unchanged from previous value

Example

<table>
<thead>
<tr>
<th>RIVRPROP</th>
<th>-- Name</th>
<th>Reaches</th>
<th>Manning Coef</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>RIVER1</td>
<td>15 17</td>
<td>0.03</td>
</tr>
<tr>
<td></td>
<td>RIVER2</td>
<td>12 12</td>
<td>0.02</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Keywords

ECLIPSE Reference Manual

1820
**RIVRXSEC  Tables of river profile data**

This keyword is used to define the cross-section of a river as a function of the depth.

The keyword is followed by \texttt{MXTBPR} tables with a maximum of \texttt{MXDPTB} entries in each. These numbers are set with the \texttt{RIVRDIMS} keyword. Each table is terminated with a slash (\texttt{/}).

Each table consists of 3 columns of data:

1. **Depth**
   
   The values should increase monotonically down the table
   
   The first entry should be 0.0
   
   **UNITS:** m (\texttt{METRIC}), ft (\texttt{FIELD}), cm (\texttt{LAB})

2. **Wetted perimeter at this depth**

   The values should increase monotonically down the table

   **UNITS:** m (\texttt{METRIC}), ft (\texttt{FIELD}), cm (\texttt{LAB})

3. **Cross-sectional area at this depth**

   The values should increase monotonically down the table

   The first entry should be 0.0

   **UNITS:** m$^2$ (\texttt{METRIC}), ft$^2$ (\texttt{FIELD}), cm$^2$ (\texttt{LAB})

There is a minimum of two rows of data per table. End the table with a slash (\texttt{/}). After the first tables subsequent tables may be defaulted to the previous table by just entering a slash.

**Example**

To define two tables for a simple box section channel 5 metres wide; the second being identical to the first:

\begin{verbatim}
RIVRXSEC
  0.0 5.0 0.0
  10.0 25.0 50.0/
/
\end{verbatim}
RIVSALT

Sets salt concentrations in the upstream flow of rivers branches

This keyword is used to specify the concentration of salt in an injection stream. Salt can be injected in the upstream flow of a river branch or in a specified reach. The keyword should only be used in runs with the brine option active (keyword BRINE in the RUNSPEC section). If the RIVSALT keyword does not appear in the SCHEDULE section, then concentration values of zero are assumed.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

1. River name
2. The concentration of salt in the injection stream.
   UNITS: kg/sm³ (METRIC), lb/stb (FIELD), gm/scce (LAB), kg/sm³ (PVT-M)
3. The branch name. If not specified, this item is defaulted to the main branch of the river, that is branch with the same name as the river.
   If the item 4 is not specified or defaulted, salt is injected in the upstream flow of this branch with the concentration specified in item 2. This should be a branch with an upstream boundary condition.
4. The reach number. If this item is specified, salt is injected in this reach with the concentration specified in item 2. item 3 is then ignored.

End data record with a slash (/).

End the set of records with a blank record, containing just a slash (/).

Example

In this example, we specify the salt concentration in the upstream flow of the main branch of RIV1. We specify the salt concentration in the upstream flow of the branch BR1 of RIV2 and inject salt in reaches number 5 and 15 of RIV2.

```
RIVSALT
RIV1 0.8 1*/
RIV2 0.8 BR1 /
RIV2 0.7 BR1 5/
RIV2 0.7 1* 15/
/```

Keywords

1822
**RIVTRACE**

Sets tracer injection concentrations for rivers

This keyword specifies the concentration of an injected tracer. If a tracer does not appear in the list of tracer names in the RIVTRACE keyword, a concentration of 0.0 is assumed.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. **River name.**
   
   Each river must previously have been declared.

2. **Name of the tracer**
   
   (Up to 3 characters)

3. **Value of the tracer concentration** $T_{\text{conc}}$
   
   If a cumulative tracer factor is specified in item 4, then $T_{\text{conc}}$ is taken as the maximum tracer concentration.

4. **The value of the cumulative tracer factor,** $T_{\text{cum}}$.
   
   The cumulative tracer factor allows the tracer concentration to be specified as a linear function of the total cumulative water flow.

   If a cumulative tracer factor is specified, then the tracer concentration in the injection stream is given by:
   
   $$T_c = \text{MIN} (CI \cdot T_{\text{cum}}, T_{\text{conc}})$$  \hspace{1cm} \text{Eq. 3.164}$$

   where

   $CI$ \hspace{0.5cm} Total cumulative water flow at the previous timestep

   $T_{\text{cum}}$ \hspace{0.5cm} Cumulative tracer factor

   $T_{\text{conc}}$ \hspace{0.5cm} Tracer concentration entered in item 3.

   **UNITS:** 1/sm$^3$ (METRIC), 1/stb (FIELD), 1/scc(LAB), 1/sm$^3$ (PVT-M)

5. **The branch name.**
   
   If no branch is specified the main branch is used.

   If item 6 is not specified or defaulted, the tracer is injected in the upstream flow of this branch with the concentration set by items 2 and 3. This should be a branch with an upstream massflow boundary condition.

6. **Reach number.**
   
   If this item is specified, the tracer is injected in this reach with the concentration set by items 2 and 3. The item 5 is then ignored.

End data record with a slash (/).

End the set of records with a blank record, containing just a slash (/).
Example

In this example tracer WT1 is being injected through the main branch of river RIV1 and through branch BR2 of river RIV2. Tracer WT2 is injected in reaches 5 and 15 of river RIV2.

```
RIVTRACE
  RIV1  WT1  1.0 3*/
  RIV2  WT1  1.0 1*  BR2  1*/
  RIV2  WT2  1.0 1*  BR2  5/ 
  RIV2  WT2  1.0 1*  1*  15/ 
/ 
```
RKPVMNMX  Specify limits for pore volume multipliers in rock tables

The data comprises NTOCC (see item 1 in keyword ROCKCOMP in the RUNSPEC section) tables of pore volume multiplier limits for hysteretic rock data, each terminated by a slash (/). Each table consists of 2 columns of data:

1. The minimum pore volume multiplier.
   UNITS: dimensionless
2. The maximum pore volume multiplier.
   UNITS: dimensionless

No default values are defined. Data must be provided for each column of a given table.

The NTOCC records correspond to the Rock Compaction table region numbers, which are set using keyword ROCKNUM (item 1).

The RKPVMNMX keyword cannot be used in THERMAL simulations. It can only be used if the Rock Compaction option is selected (keyword ROCKCOMP in the RUNSPEC section with item 1 set to the BOBERG option).

For the BOBERG model, elastic curves are provided in a tabular format using the ROCKTABH keyword, and based on these elastic curves, ECLIPSE defines the deflation and dilation curves.

Without this keyword, the dilation curve, for instance, is unbounded and so if the maximum pore volume multiplier in the ROCKTABH table is reached and the pressure continues to increase, then ECLIPSE performs a table extrapolation which can result in extremely high pore volume and transmissibility multipliers.

Use of the RKPVMNMX keyword is a way of preventing unlimited extrapolation of the deflation and dilation curves. For instance, when the maximum pore volume multiplier defined using this keyword is reached and the pressure continues to increase, the model does not dilate anymore, instead the elastic curves are used to describe the rock expansion for the current pressure.

Using the pore volume multipliers specified with this keyword, a corresponding minimum/maximum pressure is obtained in addition to a minimum/maximum transmissibility multiplier at these pressure values. If the current grid cell pressure is outside of the minimum/maximum pressure values, then a new pore volume multiplier and transmissibility multiplier is found using the elastic curves, otherwise the original ROCKTABH algorithm will be used.

See also keywords ROCKOPTS, and "Rock compaction" in the ECLIPSE Technical Description.

Example

With NTOCC = 1

| RKPVMNMX | 0.3 0.8 / |

Keywords

1825
RKTRMDIR  Allow ROCKTAB tables to contain directional transmissibility multipliers

This keyword determines how many columns the ROCKTAB tables contain. If specified, they have five columns instead of the usual three, and instead of the third column being a general transmissibility multiplier, the third column is the multiplier in the x-direction, the fourth is the multiplier in the y-direction and the fifth is the multiplier in the z-direction.

The functionality is not compatible with the NINEPOIN and MPFA keywords.

This keyword has no associated data.

Example

```
RKTRMDIR
```

Keywords

1826
ROCK compressibility

The data comprises NTPVT (see keyword TABDIMS in the RUNSPEC section) records, each terminated by a slash (/).

Each record consists of the following items of data:

1. The reference pressure ($P_{\text{ref}}$).
   
   UNITS: barsa (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)
   
   DEFAULT: 1.0132 barsa (METRIC), 14.7 psia (FIELD), 1 atm (LAB and PVT-M).

2. The rock compressibility
   
   $C = \left( \frac{dV}{dP} \right) / V$

   where $V$ is the pore volume.

   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

   ECLIPSE 100 DEFAULT: 0.0

   ECLIPSE 300 DEFAULT: 4.934E-5 1/bars (METRIC), 3.402E-6 1/psi (FIELD), 5.000E-5 1/atm (LAB and PVT-M).

   Note: If the ECLIPSE 300 Geomechanics option is specified and the second porosity-stress model is used, this rock compressibility is interpreted as $C_{pp}$.

   Geomechanics only 3. Rock matrix compressibility, $C_R$

   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

   Geomechanics only 4. Bulk compressibility, $C_{bc}$

   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

   Geomechanics only 5. Reference porosity $\phi_0$ for estimating compressibility if items 3 or 4 are unset.

   UNITS: m$^3$/m$^3$ (METRIC), ft$^3$/ft$^3$ (FIELD), cc/cc (LAB), m$^3$/m$^3$ (PVT-M)

   Geomechanics only 6. Reference Poisson’s ratio $\nu_R$ for estimating compressibility if item 4 is unset.

   ECLIPSE adjusts the pore volumes using

   $V(P) = V(P_{\text{ref}}) \left( 1 + X + \frac{X^2}{2} \right)$  \hspace{1cm} \text{Eq. 3.165}$

   where $X = C (P - P_{\text{ref}})$

   and $V(P_{\text{ref}})$ is the pore volume at the reference pressure.

   The NTPVT records correspond to the PVT region numbers, which are set using keyword PVNUM.

   If item 3 of the ROCKOPTS keyword is set to the string 'SATNUM', the ROCK data records associate with saturation function region numbers (set with keyword SATNUM) instead of PVT region numbers. In this case, the ROCK keyword data should comprise NTSFUN (see keyword TABDIMS) records, corresponding to the saturation region numbers.
If item 3 of the ROCKOPTS keyword is set to the string 'ROCKNUM', the ROCK data records associate with rock region numbers (set with keyword ROCKNUM) instead of PVT region numbers. In this case, the ROCK keyword data should comprise NTROCC (see keyword TABDIMS) records, corresponding to the rock region numbers.

The ROCK keyword must be used if the Rock Compaction option is not selected (keyword ROCKCOMP in the RUNSPEC section). If Rock Compaction is selected, keyword ROCKTAB should be used instead.

See keyword ROCKTAB for details of this option.

**Note:** In ECLIPSE, \( V(P_{ref}) \) is taken to be the value of \( PORV \) which is calculated from the geometric information supplied in the GRID and EDIT sections. Thus \( PORV \) (and \( PORO \)) values must be regarded as being at the rock reference pressure. It should be borne in mind that other simulators will assume the geometric \( PORV \) represents reservoir pore volumes. The reservoir pore volume in ECLIPSE is calculated as \( V(p) \) as above.

---

**ECLIPSE 300**

Setting item 2 of the ROCKOPTS keyword to 'STORE' implies that the initial equilibrated pressures are used as reference pressures in each grid block instead of the value of \( P_{ref} \) entered in the ROCK keyword.

Then the input pore volume for a cell is defined as the reservoir pore volume at initial conditions rather than the pore volume at the reference pressure.

**Note:** If ECLIPSE 300 Geomechanics is specified together with the second porosity-stress relationship model, which is activated in the 4th item of keyword GEOMECH, then the following relationships are used. If the bulk compressibility, \( C_{bc} \) is unset in item 4 above, then \( C_{bc} = \phi_0 \cdot C_{pp} + (1 + \phi_0 \cdot )C_R \) where \( C_{pp} \) is entered as item 2 above. If the rock matrix compressibility \( C_R \) is also unset in item 3 above, then \( C_R \) is estimated from the Hashin-Shtrikman bounds to be \( C_R = \frac{C_{bc}}{1 + \frac{3(1-\nu_R)\phi_0}{2(1-2\nu_R)(1-\phi_0)}} \) if the reference Poisson ratio \( \nu_R \) is set in item 6 or \( C_R = \frac{C_{bc}}{1 + \frac{3\phi_0}{2(1-\phi_0)}} \) if it is not. Note that if \( C_{pp} \) is unset, then a default value of 0.00005 bars\(^{-1}\) will be assumed. Also note that if the reference porosity \( \phi_0 \) is not entered in item 5, then a warning is generated and a value of 0.33 is assumed. Further information on these estimates can be found in Zimmerman, [Ref. 134]. You are responsible for ensuring that the above estimates for any defaulted compressibilities are reasonable.

**Note:** The rock formation volume factor \( B_r \), used to relate rock volumes at reservoir conditions to standard conditions, is calculated from item 2 through \( C_r = \left( \frac{dB_r}{dP} \right) \mid_{B_r} \).

---

**Example**

With NTPVT=3:

```
ROCK
3600.00 .40E-05 /
```

---

**Keywords**
<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>3600.00</td>
<td>0.40E-05</td>
<td>/</td>
</tr>
<tr>
<td>3000</td>
<td>0</td>
<td>/</td>
</tr>
</tbody>
</table>

Keywords

1829
ROCK2D 2D rock compaction tables versus P and Sw

The data comprises NTROCC (see keyword ROCKCOMP in the RUNSPEC section) tables of water induced compaction data, in the form of two-dimensional tables of pore volume multiplier versus pressure and water saturation increase.

Each table consists of up to NSSFUN (see keyword TABDIMS in the RUNSPEC section) records of data, and is terminated by a slash (/). Each record consists of up to NSSFUN data items, terminated by a slash, defining the two-dimensional function. Each record consists of the following data:

Item 1 The pressure value.

   The pressure values must increase monotonically within the set of records defining a table.

   UNITS: barsa (METRIC), psia (FIELD), atma (LAB)

Subsequent items The pore volume multiplier corresponding to the pressure in item 1 and each saturation increase value in the corresponding ROCKWNOD table.

   Successive values within a record should be level or decreasing, and the number of entries should be equal to the number of entries in the corresponding ROCKWNOD table.

   The NTROCC tables correspond to the rock compaction table region numbers, which are set using keyword ROCKNUM.

The ROCK2DTR keyword can optionally be used in conjunction with ROCK2D to specify the associated transmissibility modification as a function of both pressure and water saturation. If the ROCK2DTR keyword is absent the transmissibility is kept constant.

The behavior of the reservoir when the pressure in a grid block increases will depend on the first data item in the ROCKCOMP keyword in the RUNSPEC section. If the reversible option is selected the two-dimensional table will be looked up at the prevailing pressure. If the irreversible option is selected the table will be looked up using the minimum pressure that the grid block attains during the run.

If the overburden pressure supplied in the OVERBURD keyword is greater than the fluid pressure (which it often will be), the effective fluid pressure \( P_{\text{eff\,fl}} = P_{\text{fluid}} - \text{Overburden} \) will be negative. In this case the ROCK2D table will have to be supplied with negative pressures in column 1. At present the STRESS option in the ROCKOPTS keyword is not available with ROCK2D.

See also keywords ROCKTABW, ROCK2DTR, ROCKWNOD, ROCKOPTS and OVERBURD, and "Rock compaction" in the ECLIPSE Technical Description.

Example

With NTROCC=1 and NSSFUN≥4:

<table>
<thead>
<tr>
<th>P (barsa)</th>
<th>Sw</th>
<th>Multiplier</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.88</td>
<td>0.88</td>
</tr>
<tr>
<td></td>
<td>0.88</td>
<td>0.88</td>
</tr>
<tr>
<td></td>
<td>0.88 /</td>
<td></td>
</tr>
<tr>
<td>1000.0</td>
<td>0.92</td>
<td>0.92</td>
</tr>
<tr>
<td></td>
<td>0.92</td>
<td>0.90</td>
</tr>
<tr>
<td></td>
<td>0.90 /</td>
<td></td>
</tr>
<tr>
<td>9000.0</td>
<td>1.0</td>
<td>1.0</td>
</tr>
</tbody>
</table>

Keywords

1830
0.94
0.94 /
/
ROCK2DTR  2D transmissibility modifier tables for water induced rock compaction

The ROCK2DTR keyword can optionally be used in runs where the water induced compaction model has been activated by the ROCK2D keyword, to specify the associated transmissibility modification as a function of both pressure and water saturation. If the ROCK2DTR keyword is absent (but ROCK2D is present) the transmissibility is kept constant.

The data comprises NTROCC (see keyword ROCKCOMP in the RUNSPEC section) tables of water induced compaction transmissibility multipliers, in the form of two-dimensional tables versus. pressure and water saturation increase.

Each table consists of up to NSSFUN (see keyword TABDIMS in the RUNSPEC section) records of data, and is terminated by a slash (/). Each record consists of up to NSSFUN data items, terminated by a slash, defining the two-dimensional function. Each record consists of the following data:

Item 1    The pressure value.

The pressure values must increase monotonically within the set of records defining a table.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB)

Subsequent items: The transmissibility multiplier corresponding to the pressure in item 1 and each saturation increase value in the corresponding ROCKWNOD table.

The number of entries should be equal to the number of entries in the corresponding ROCKWNOD table.

The NTROCC tables correspond to the rock compaction table region numbers, which are set using keyword ROCKNUM.

The behavior of the reservoir when the pressure in a grid block increases will depend on the first data item in the ROCKCOMP keyword in the RUNSPEC section. If the reversible option is selected the two-dimensional table will be looked up at the prevailing pressure. If the irreversible option is selected the table will be looked up using the minimum pressure that the grid block attains during the run.

If the overburden pressure supplied in the OVERBURD keyword is greater than the fluid pressure (which it often will be), the effective fluid pressure ($P_{\text{eff}} = P_{\text{fluid}} - \text{Overburden}$) will be negative. In this case the ROCK2D table will have to be supplied with negative pressures in column 1. At present the STRESS option in the ROCKOPTS keyword is not available with ROCK2D.

See also keywords ROCKTABW, ROCK2D, ROCKWNOD, ROCKOPTS and OVERBURD, and "Rock compaction" in the ECLIPSE Technical Description.

Example

With NTROCC=1 and NSSFUN ≥4:

<table>
<thead>
<tr>
<th>ROCK2DTR</th>
<th>0.0</th>
<th>0.81</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.80</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.74</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.73</td>
<td>/</td>
</tr>
<tr>
<td>1000.0</td>
<td>0.93</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.92</td>
<td></td>
</tr>
</tbody>
</table>

Keywords

1832
<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0.87</td>
<td>0.86</td>
<td>/</td>
<td>9000.0</td>
<td>1.0</td>
</tr>
<tr>
<td></td>
<td>1.0</td>
<td></td>
<td>1.0</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.94</td>
<td></td>
<td></td>
<td>0.94</td>
</tr>
</tbody>
</table>

Keywords
ROCKCOMP Enables the rock compaction option

This indicates that the rock compaction option is to be used (see "Rock compaction" in the ECLIPSE Technical Description). Pressure dependent pore volume and transmissibility multipliers should be entered using the keyword ROCKTAB (or ROCKTABH for hysteretic rock compaction) in the PROPS section. For cases where the Palmer-Mansoori option is being used, the rock model data must be entered using ROCKPAMA, ROCKPAME or ROCKPMAT.

The keyword may be followed by the following items to specify whether rock compaction is irreversible or hysteretic, to supply the dimensions for rock compaction tables and to initiate the water induced compaction option. If the water induced compaction model is requested in data item 3, either the ROCKTABW or the ROCK2D and ROCKWNOD keywords should be entered in the PROPS section.

The data should be terminated by a slash (/).

1. Option for rock compaction hysteresis or irreversibility.
   - REVERS Rock compaction is fully reversible with increasing pressure.
   - IRREVERS Rock compaction is irreversible; the pore space does not reinflate when the pressure increases.
   - HYSTER Rock compaction is hysteretic; the pore space deflates and reinflates elastically for pressures above an elastic limit given by the deflation curve. For cells on the deflation curve, the pore spaces will only reinflate partially when the pressure increases. Pressure-dependent pore volume and transmissibility multipliers should be entered using keyword ROCKTABH in the PROPS section.
   - BOBERG Rock compaction is hysteretic; the pore space deflates and reinflates elastically for pressures above an elastic minimum given by the deflation curve and below an elastic maximum given by the dilation curve. For cells on the deflation curve, the pore spaces will only reinflate partially when the pressure increases. For cells on the dilation curve, the pore spaces will only deflate partially when the pressure decreases. Pressure-dependent pore volume and transmissibility multipliers should be entered using keyword ROCKTABH in the PROPS section.

   ECLIPSE 100 only
   - REVLIMIT This option is only available if the water induced compaction model is active (data item 3). Compaction is reversible up to a limit of the pore volume associated with the minimum pressure reached in a grid block and the initial water saturation. The aim is to model elastic/plastic behavior with only the elastic region being reversible.

   PALM–MAN This option is only available with the Coal Bed Methane option (see COAL). The Palmer-Mansoori model ([Ref. 78] in the ECLIPSE Technical Description) is used to calculate the pore volume and transmissibility multipliers. The parameters for the Palmer-Mansoori model should be entered using keyword ROCKPAMA in the PROPS section.

   ECLIPSE 300 only
   - TPLM–MAN This option is only available with the THERMAL option (see THERMAL). The Thermal Palmer-Mansoori model ([Ref. 78] in the ECLIPSE Technical Description) is used to calculate the pore volume and transmissibility multipliers. The parameters for the Palmer-Mansoori model should be entered using keyword ROCKPMAT in the PROPS section.
NONE

Turns off rock compaction. This is equivalent to not specifying the ROCKCOMP keyword, unless the water induced compaction model has been activated.

DEFAULT: REVERS

2. The number of rock compaction tables.

The number of tables (NTROCC) entered using ROCKPAMA, ROCKPMAT, ROCKTAB or ROCKTABH in the PROPS section.

**ECLIPSE 300**

The number of tables may also be entered using item 13 of the TABDIMS keyword. If both ROCKCOMP item 2 and TABDIMS item 13 are present, the greater of the two values is used.

Different rock compaction tables may be used in different parts of the reservoir if the keyword ROCKNUM is specified in the REGIONS section. The maximum number of pressure nodes in the table is specified by NPPVT (item 4 of TABDIMS).

DEFAULT: 1

**ECLIPSE 100 only**

3. Flag requesting the water induced compaction option.

YES Use the water induced compaction option

NO Do not use the water induced compaction option

DEFAULT: NO

**ECLIPSE 300**

4. Option to define a model which can be used to specify the dependence of transmissibility on porosity:

EXP Transmissibility is as exponential function of porosity. Please refer to keywords, ROCKTRMX, ROCKTRMY and ROCKTRMZ for a fuller description.

CZ Transmissibility is a function of porosity via the Carmen-Kozeny type formula:

\[ \frac{\kappa}{\kappa_0} = \frac{\phi}{\phi_0} \left[ \frac{1 - \phi_0}{1 - \phi} \right]^\tau \]

where \( \tau \) is defined in item 5 below.

item 4 is available for use with the ROCKTAB and ROCKTABH keywords. It is currently not available for use with the ROCK keyword.

**ECLIPSE 300**

5. The exponent (\( \tau \)) in the Carmen-Kozeny formula defined using item 4 above. The lower limit is 0, the upper limit is 10 and values are restricted to the integers.

DEFAULT: 0

**Example**

Irreversible rock compaction with 2 rock compaction tables and no water induced compaction

```
ROCKCOMP: IRREVERS  2 /
```
ROCKCON  Connection data for cap and base rocks

This keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section).

ROCKCON connects cap and base rocks (declared using ROCKPROP) to one or more reservoir cells. The connection to the reservoir is set up by an arbitrary box defined with lower and upper I, J and K indices (refer to items 2-7 below). For each active grid block within the box, a rock connection is made to the face specified in item 8 if either of the following conditions are met:

- The face is on the outside of the reservoir, or
- The face adjoins an inactive cell.

Any number of records may be entered, each terminated with a slash (/). The set of records must be terminated with a blank record containing just a slash (/).

The items in each record are:

1. Rock type identification number
   This should be a number between 1 and the maximum number of rocks, entered using ROCKDIMS in the RUNSPEC section.

2. Lower I-location of connecting grid blocks

3. Upper I-location of connecting grid blocks

4. Lower J-location of connecting grid blocks

5. Upper J-location of connecting grid blocks

6. Lower K-location of connecting grid blocks

7. Upper K-location of connecting grid blocks

8. Descriptor defining the face of the reservoir to which the rock connects (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS):
   
   I- or X-  I or X     = 1   face
   
   I+ or X+  I or X     = NDIVIX face
   
   J- or Y-  J or Y     = 1   face
   
   J+ or Y+  J or Y     = NDIVIY face
   
   K- or Z-  K or Z = 1   face (top)
   
   K+ or Z+  K or Z = NDIVIZ face (bottom)

9. The rock influx coefficient.
   This is a transmissibility multiplier for the connection between the rock and grid.
   DEFAULT: 1.0

   The effect of the multiplier is cumulative. If the same block is specified twice with coefficients of 0.5, the final coefficient is 0.25.

The rock connections can be listed by specifying the ROCKCON mnemonic in the RPTSol keyword.
An array with the name ROCKCONA is written to the INIT file to illustrate which cells are connected to each rock. If a cell is connected to rock N then the number \(2^{(N-1)}\) appears in the ROCKCONA array. If a cell is connected to multiple rocks then the appropriate output numbers are summed.

**Note:** Each unique rock can only be connected to one grid. In other words, a rock can only be connected to either the global grid or to a single local grid refinement.

See "Thermal option" in the ECLIPSE Technical Description for further information on this option.

**Example**

Two rock types, connected to the top and bottom of a 10 x 10 x 2 grid, with no rock influx coefficient.

```
ROCKCON
1 1 10 1 10 1 1 'K-' /
2 1 10 1 10 2 2 'K+' /
/
```

Keywords

1837
**ROCKCONL**  Specifies local grid refinement connection data for cap and base rocks

This keyword can only be used with the THERMAL option (keyword **THERMAL** in the RUNSPEC section).

ROCKCONL connects cap and base rocks (declared using **ROCKPROP**) to one or more reservoir cells.

The connection to the reservoir is set up by an arbitrary box defined with lower and upper I, J and K indices within the local grid refinement (refer to items 3-8 below). For each active grid block within the box, a rock connection is made to the face specified in item 9 if either of the following conditions are met:

- The face is on the outside of the reservoir, or
- The face adjoins an inactive cell.

Any number of records may be entered, each terminated with a slash (/). The set of records must be terminated with a blank record containing just a slash (/).

The items in each record are:

1. Rock type identification number
   
   This should be a number between 1 and the maximum number of rocks, entered using **ROCKDIMS** in the RUNSPEC section.

2. Local grid refinement name (up to 8 characters).

3. Lower I-location of connecting grid blocks.

4. Upper I-location of connecting grid blocks.

5. Lower J-location of connecting grid blocks.


7. Lower K-location of connecting grid blocks.


9. Descriptor defining the face of the reservoir to which the rock connects (where **NDIVIX**, **NDIVIY** and **NDIVIZ** are specified in RUNSPEC keyword **DIMENS**):

   - I- I = 1 of LGR face
   - I+ I = NDIVIX of LGR face
   - J- J = 1 of LGR face
   - J+ J = NDIVIY of LGR face
   - K- K = 1 of LGR face (top)
   - K+ K = NDIVIZ of LGR face (bottom)

10. The rock influx coefficient.

   This is a transmissibility multiplier for the connection between the rock and grid.

   DEFAULT: 1.0
The effect of the multiplier is cumulative. If the same block is specified twice with coefficients of 0.5, the final coefficient is 0.25.

The rock connections to local grid refinements can be listed by specifying the ROCKCON mnemonic in the RPTSOI keyword.

An array with the name ROCKCONA is written to the INIT file to illustrate which cells are connected to each rock. If a cell is connected to rock \( N \) then the number \( 2^{(N-1)} \) appears in the ROCKCONA array. If a cell is connected to multiple rocks then the appropriate output numbers are summed.

**Note:** Each unique rock can only be connected to one grid. In other words, a rock can only be connected to either the global grid or to a single local grid refinement.

See "Thermal option" in the ECLIPSE Technical Description for further information on this option.

**Example**

Two rock types, connected to the top and bottom of a 10 x 10 x 2 local grid refinement (called LGRROCK), with no rock influx coefficient.

```
ROCKCONL
1 LGRROCK 1 10 1 10 1 1 'K-' /
2 LGRROCK 1 10 1 10 2 2 'K+' /
```

Keywords
1839
ROCKDEN

Rock density

This keyword has three uses. It can be used with the Geomechanics option (keyword GEOMECH) for rock stress balance calculation, or with runs with solids (keyword SOLID) for adsorption calculations, or with runs with the coal bed methane option (keyword COAL) for specifying the coal density. If neither GEOMECH, SOLID or COAL is specified in the RUNSPEC section this keyword has no effect.

The keyword should be followed by one non-negative real number for every grid block in the current input box, specifying the rock density. The data must be terminated by a slash (/).

UNITS: kg/m$^3$ (METRIC), lb/ft$^3$ (FIELD), gm/cc (LAB), kg/m$^3$ (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces may not be inserted on either side of the asterisk.

Note: The default for rock density is zero. If this keyword is not specified, the simulation proceeds but in the Geomechanics option the gravity body forces are zero.

Example

ROCKDEN 324*128 /

Keywords

1840
ROCKDIMS  Rock over and underburden dimensions

This keyword can only be used with the THERMAL option (see "Thermal option" in the ECLIPSE Technical Description).

The data consists of the following items of data, describing the number of types of cap and base rock to be used in the run to model heat loss or gain from the reservoir.

1. The maximum number of rock types
2. Not used
3. The maximum number of lines of connection data
   DEFAULT: The maximum number of rock types.

Example

Two rock types, defined using two lines of ROCKCON and/or ROCKCONL data for each type.

ROCKDIMS
2 1* 4 /
**ROCKFRAC  Rock volume fraction**

The keyword is used to calculate the rock volumes. For each grid cell the rock volume is calculated as the bulk volume times the fractional value. The keyword is optional.

The keyword should be followed by one non-negative real number for every grid block in the current input box, specifying the fractional value. The data must be terminated by a slash (/).

This keyword will typically be useful in dual/multi-porosity runs with:

- The **THERMAL** option, where energy is stored in the rock
- The **COAL** option (shale gas) where the rock contains adsorbed gas.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0.9). Note that spaces may not be inserted on either side of the asterisk.

If the keyword is defaulted, the standard procedure for calculating rock volumes will be applied.

**ECLIPSE 100 only**

Keyword **ROCKFRAC** is not available in runs with the triple porosity (**TRPLPORO**) or discretized matrix (**NMATRIX**) options.

UNITs: dimensionless

**Example**

This defines all cell values for a grid of size 10 x 10 x 10 (5 layers in each of 2 porosities). More rock is put into the matrix than into the fracture.

```plaintext
ROCKFRAC
500*0.8 500*0.1 /
```
ROCKNUM  Rock compaction table region numbers

The keyword should be followed by one integer for every grid block in the current input box, specifying the rock compaction table region to which it belongs. The region number should not be less than 1 or greater than NTROCC (see keyword ROCKCOMP). The data must be terminated by a slash (/).

The region number specifies which rock compaction table (input using ROCKPAMA, ROCKTAB, ROCKTABH, ROCKTABW or ROCK2D and ROCK2DTR in the PROPS section) should be used to calculate the porosity and permeability multipliers for each grid block. The Rock Compaction option must be enabled using keyword ROCKCOMP.

Note: ROCKNUM can also be used together with the ROCK keyword for specifying rock compressibility. See keywords ROCK and ROCKOPTS.

If the keyword ROCKSIG or ROCKTHSG is used to specify pressure tables of the sigma multiplier for matrix-fracture coupling in dual porosity runs, only the ROCKNUM numbers of the matrix cells are used to obtain the pressure tables. The ROCKNUM numbers of the fracture cells are ignored.

The rock compaction region numbers can be modified during the simulation by re-specifying the ROCKNUM keyword in the SCHEDULE section. This procedure should only be carried out with care and is not recommended in general. It cannot be used in a parallel run, and changes to ROCKNUM will not be honored in a restart run unless the re-specification is also included in the restart data file.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

Note: If ROCKNUM is unspecified for a cell, the default value for ROCKNUM will be based on the ROCKOPTS keyword item 3. If item 3 is set to PVTNUM, then the defaulted value will be based on PVTNUM; if item 3 is set to SATNUM, then it will be based on SATNUM; and if item 3 is set to ROCKNUM, then the defaulted value of 1 will be used.

Example

With NTROCC=4 specified in keyword ROCKCOMP; NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS; and no input BOX set:

ROCKNUM
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4
8*1 16*2 8*3 16*4 /
ROCKOPTS  Options for rock compaction or rock compressibility

The ROCKOPTS keyword is used to control options associated with the rock compaction and compressibility facilities. It should be followed by some or all of the following data items, terminated by a slash (/).

1. The method of application of OVERBURD to the tabulated pressures in the ROCKTAB, ROCKTABH keywords:

   OPTIONS: PRESSURE or STRESS

   If the overburden pressure supplied in the OVERBURD keyword is greater than the fluid pressure (which it often will be), the effective fluid pressure \( P_{\text{eff,fl}} = P_{\text{fluid}} - \text{Overburden} \) will be negative. In this case the rock compaction tables will have to be entered with negative pressure values. The tables can be input in a more intuitive manner by selecting the STRESS option before the tables are entered. In this case the pore volume and transmissibility multipliers will be tabulated against the effective overburden pressure, \( P_{\text{eff,ov}} = \text{Overburden} - P_{\text{fluid}} \), which will then be positive.

   This item should be left defaulted if the OVERBURD keyword is not used.

   DEFAULT: PRESSURE

2. The reference pressure option:

   OPTIONS: STORE or NOSTORE

   The STORE option copies the initial equilibrated pressure at the start of the run into the overburden array. This has the effect of referencing the pore volume (as a function of pressure) to the initial pressure rather than the reference pressure, that is, the pressure with a pore volume multiplier of 1.0 in the ROCKTAB or ROCKTABH tables. Thus the input pore volume for a cell is defined as the pore volume at initial conditions rather than the pore volume at the reference pressure.

   Note that the OVERBURD keyword should not be used with the STORE option as the input OVERBURD data will be overwritten and ignored.

   DEFAULT: NOSTORE

3. Table region to be used:

   Possible values are: ROCKNUM, SATNUM or PVTNUM

   DEFAULT: PVTNUM

   If SATNUM or ROCKNUM are not found in the REGIONS section, default PVTNUM values will be used. Note that the number of ROCK tables must be set with TABDIMS item 13.

4. Flag determining the initial conditions for the HYSTER and BOBERG options.

   OPTIONS:

   DEFLATION  The reservoir rock is assumed to be fully compacted and initial pore volume and transmissibility multipliers are calculated from the deflation curve.
ELASTIC  The reservoir rock is assumed to lie on an elastic curve with pore volume multiplier equal to 1 at the cell pressure.

*Note:* The elastic curve will not be calculated if the initial pressure is below the minimum pressure in the table. If the initial pressure is below the minimum pressure in the table the deflation curve will be used and this could lead to unphysical values. It is recommend to ensure that the initial pressure is within the specified range of pressure in the table(s).

**DEFAULT:** DEFLATION

See also keywords ROCKTAB, ROCKTABH, ROCK2D, ROCK2DTR and OVERBURD, and "Rock compaction" in the *ECLIPSE Technical Description*.

*Note:* If ROCKNUM is unspecified for a cell, the default value for ROCKNUM will be based on the ROCKOPTS keyword item 3. If item 3 is set to PVNUM, then the defaulted value will be based on PVNUM; if item 3 is set to SATNUM, then it will be based on SATNUM; and if item 3 is set to ROCKNUM, then the defaulted value of 1 will be used.

**Example**

```plaintext
ROCKOPTS
  STRESS NOSTORE/
```

**Keywords**

1845
ROCKPAMA Palmer-Mansoori rock model data

This keyword is only available in the coal bed methane option (see keyword COAL in the RUNSPEC section). It is used to specify data for the Palmer-Mansoori rock model ("Palmer-Mansoori rock model" in ECLIPSE Technical Description).

The data comprises NTROCC (see keyword ROCKCOMP in the RUNSPEC section) rows of Palmer-Mansoori rock model data, each terminated by a slash (/). If a row is left blank, then its values are copied from the previous row.

Each row consists of the following columns of data:

1. The bulk modulus, $K$  
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
2. The constrained axial modulus, $M$  
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
3. Grain compressibility, $\gamma$  
   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)  
   DEFAULT: 0
4. Langmuir curve parameter, $\beta$  
   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)  
   DEFAULT: 0
5. Langmuir curve parameter, $\epsilon_l$  
   UNITS: dimensionless  
   DEFAULT: 0
6. Fraction, $f$, between 0 and 1  
   UNITS: dimensionless  
   DEFAULT: 0.5
7. Exponent in the porosity-permeability relationship, $n$  
   UNITS: dimensionless  
   DEFAULT: 3
8. Geometric factor, $g$  
   UNITS: dimensionless  
   DEFAULT: 1.0
9. Langmuir solvent curve parameter, $\beta_s$  
   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)  
   DEFAULT: Same as item 4, $\beta$
10. Langmuir solvent curve parameter, $\epsilon_s$
The Palmer-Mansoori model adjusts the pore volumes, $V(P)$, at the current pressure, $P$, using

$$V(P) = V(P_0) \left[ 1 + \frac{c_m}{\varphi_0} (P - P_0) + \frac{1}{\varphi_0} \left( \frac{K}{M} - 1 \right) \left( \epsilon_{ps} - \epsilon_{ps,0} \right) \right]$$

where

- $\varphi_0$ is the initial porosity,
- $V(P_0)$ is the pore volume at the initial pressure, $P_0$, and
- $c_m = \frac{g}{M} - \left[ \frac{K}{M} + f - 1 \right] \gamma$.

Following Palmer-Mansoori, assuming that permeability varies with porosity as

$$\frac{k}{k_0} = \left( \frac{\varphi}{\varphi_0} \right)^n$$

then the transmissibility multiplier is given by

$$\left[ 1 + \frac{c_m}{\varphi_0} (P - P_0) + \frac{1}{\varphi_0} \left( \frac{K}{M} - 1 \right) \left( \epsilon_{ps} - \epsilon_{ps,0} \right) \right]^n$$

The calculation of the volumetric strain term due to gas adsorption or desorption ($\epsilon_{ps}$) depends on the setting of `CBMOPTS` item 5, where $\epsilon_{ps,0}$ is the initial strain term.

**PMSTD:**

This is the standard form assuming instant adsorption

$$\epsilon_{ps}(P) = \frac{\epsilon_l \beta P}{1 + \beta P}$$

The pre-2013.1 version used modifications of the derivative of the pore volume multiplier with respect to pressure, for negative derivatives. This is removed from the 2013.1 release and onwards. See item 179 in `OPTIONS3` and item 96 in `OPTIONS` for an explanation of the old behavior. The pre-2013.1 method can be activated by using item 295 in `OPTIONS3` or item 200 in `OPTIONS`.

It is recommended that you investigate the other models described below if the standard model is not appropriate and the simulator is struggling. This will be the case if the pore volume is changing due to the volumetric strain term ($\epsilon_{ps}$) without a corresponding exchange of gas between the fracture pore volume and the coal matrix.

**PMADTN or PMADIMP:**

For cases where the diffusion process is much slower than the assumed instant adsorption of the standard method, these methods are preferred. For ECLIPSE 300 these methods require `TPAMEPS`. The standard method would in these cases suffer from an incorrect strain term that does not correspond to the desorption/adsorption process. This could lead to a negative total compressibility and the reservoir equations could become unsolvable. To overcome this the strain term is formulated as:

$$\epsilon_{ps}(G_C, S_C) = \frac{\epsilon_l \beta_g P_g + \epsilon_s \beta_s P_s}{1 + \beta_g P_g + \beta_s P_s}$$
where

\( P_g(G_C) \) is the inverse look-up of the equilibrium gas pressure, corresponding to the adsorbed gas \( G_C \) and

\( P_s(S_C) \) is the equilibrium solvent pressure of the adsorbed solvent concentration \( S_C \).

Here

\( G_C \) is the bulk gas concentration in the coal matrix (surface volume / unit coal volume)

\( S_C \) is the bulk solvent concentration in the coal matrix (surface volume / unit coal volume).

These matrix coal gas concentration quantities can be output by \textsc{RPTRST} using \textsc{COALGAS} or \textsc{RPTSCHED} using \textsc{GASCONC}. Using the \textsc{LANGMUIR} and \textsc{LANGSOLV} input, the pressures \( P_g \) and \( P_s \) can be obtained from the corresponding pressure entry of the coal gas concentration within these tables. These pressures are the equilibrium pressure for the single component, gas or solvent, at the given bulk coal gas concentrations,

\[ G_C = \text{LANGMUIR}(P_g) \quad \text{and} \quad S_C = \text{LANGSOLV}(P_s). \]

Note that using the different sorption models \textsc{COALADS} or \textsc{COALPP} also affects the computations of the \( \varepsilon_p \) term, since the adsorption models are different.

Alternatively the \( \varepsilon_p \) term can be looked up directly from the \textsc{TPAMEPS} and \textsc{TPAMEPSS} tables (ECLIPSE 100 and ECLIPSE 300)

\[ \varepsilon_p(G_C, S_C) = T_{\text{PAMEPS}}(G_C) + T_{\text{PAMEPSS}}(S_C) \]

which enables the user to specify the sorption-induced strain term directly as a function of the adsorbed quantities. If \textsc{TPAMEPS} is used and the case includes \textsc{SOLVENT}, \textsc{TPAMEPSS} should also be present, otherwise the solvent concentration is not accounted for (\( \varepsilon_s \) is not used).

For ECLIPSE 300, \textsc{TPAMEPS} tabulates the \( \varepsilon_p \) term for each \textsc{NCOMPS} component, where the total volumetric strain is the sum over each component:

\[ \varepsilon_p(G_{c_1}, \ldots, G_{c_{\text{NCOMPS}}}) = \left( \sum_{i=1}^{\text{NCOMPS}} T_{\text{PAMEPS}}(G_{c_i}) \right) \]

The bulk coal gas concentration can be output by using \textsc{RPTRST} or \textsc{RPTSCHED} using \textsc{CGAS}.

Using the \textsc{PMADIMP} method, the fracture pore volume derivatives with respect to the matrix simulation grid cells unknowns are included in the Jacobian matrix of the reservoir equations, but are ignored for the transmissibility multipliers. The \textsc{PMADTN} method uses the adsorbed gas in the matrix for the sorption-induced swelling/shrinkage strain at the current timestep, given by the Newton iterations. However, derivatives with respect to the gas concentrations are not included in the Jacobian matrix.

\textsc{PMADTT} This method is doing exactly the same as \textsc{PMADTN} but use the gas concentration and solvent concentrations at the previous timestep. For ECLIPSE 300 this method requires \textsc{TPAMEPS}.

\textbf{Note}: The alternative methods will only equal the standard Palmer-Mansoori method (\textsc{PMSTD}) when an equilibrium is reached (and when used with a single component gas). If the diffusion process is slow so that the adsorption capacity and the adsorbed bulk gas concentration differ a lot, the transient response of the standard method will predict a sorption induced strain as a function of the fracture pressure only, and not...
capture the time lag of the gas sorption process. It is recommended that you evaluate whether the diffusion process will have a significant impact on the rock modeling by comparing the PMSTD method with the alternative models before selecting the standard method for use.

Note: If the diffusion process is fast it is recommended that you do not use the PMADTT method because the approximation using the bulk gas concentration at the previous timestep could lead to inaccurate results.

With tabular input, it is important to include enough data points to achieve accurate modeling, but for many cases, the sorption induced strain term will be a linear function with respect to the adsorbed concentration. However, pay attention to the end points of the table as this will become the maximum value. The minimum value is given in the first row of the table and should usually be zero.

For ECLIPSE 100 cases, having a scaling of the adsorption capacities using MLANG, MLANGSLV or GASSATC, the corresponding strain terms $\varepsilon_1$, $\varepsilon_2$ are scaled with the same value as the LANGMUIR and LANGSOLV curves. This can be disabled using item 6 of CBMOPTS.

In ECLIPSE 300 the LANGMULT and GASSATC values are used if present, otherwise an average of the LANGMULC or GASADEC component specific values are used. For the compositional case, it might be better to supply the strain by using TPAMEPS values, where the scaling is not required and the individual components strain term can be specified.

Summary output of the multiplier for a given grid block is available through the keywords BPORVMOD and BPERMMOD in the SUMMARY section. The output to restart can be obtained by using RPTRST with ROCKC.

Notes

- The Palmer-Mansoori rock compaction model assumes that the input porosities are taken as the porosities at reservoir conditions.
- The rock model only applies to the pore-volume grid-cells ($\text{COALNUM} = 0$).
- Only the first part of the equation, the compression term, is used if the pore volume is not connected to coal. See keyword COALNUM.
- CBMOPTS item 4 can be used to activate monitoring of passes through the rebound region (when the pore volume pressure derivative changes sign). This enables the output of the number of passes through the rebound region using mnemonic NPMREB in RPTSCHED or RPTRST.
- CBMOPTS item 5 can be used to set a different method from the standard Palmer-Mansoori method. This method captures the time dependence of the strain term when the sorption time is controlling the exchange of gas from the matrix.
- CBMOPTS item 6 can be used to deactivate scaling of the strain term.
- CBMOPTS item 7 can be used to select that the method only apply to the transmissibility or the pore volume, or to apply a compression term to the pore volume but use the multiplier for the transmissibility.
- CBMOPTS item 8 can be used to select pressure chop when passing through the rebound pressure (ECLIPSE 100)
- CBMOPTS item 9 can be used to set a minimum pore volume to be used if the pore volume multiplier results in a pore volume being below this value.

Keywords

1849
Example

In FIELD units, and with NTROCC=3 specified in keyword ROCKCOMP:

```
ROCKPAMA
450000  600000  0  0.0016  0.0128  /
500000  700000  0  0.0010  0.0080  /
450000  600000  /
```
**ROCKPAME  Compositional coal swelling and shrinkage**

This keyword is only available in the coal bed methane option (see keyword COAL in the RUNSPEC section). It is used to specify data for a modified Palmer-Mansoori rock model. See also keyword ROCKPAMA.

A similar approach is described in [Ref. 73]. An important difference is that the model described here will not predict volume changes for undersaturated coal.

The data comprises NTCREG records of input data as specified by item 6 of REGDIMS in the RUNSPEC section and applies to the different coal region numbers as assigned by COALNUM. Each record is terminated by a slash (/) and the keyword is terminated by a final slash (/).

The data follows the Palmer-Mansoori rock model data, except that each component are assigned Langmuir curve parameters. If a row is left blank, then its values are copied from the previous row.

Note that CBMOPTS items 4, 6, 7 and 9 apply to this keyword.

For cases where the instant adsorption assumption in this model is far from valid, the ROCKPAMA model together with TPAMEPS should be considered.

Each row consists of $5 + 2 \times nc$ columns of data, where $nc$ is the number of components as set by the keyword NCOMPS.

**Item 1**
The bulk modulus, $K$

UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

**Item 2**
The constrained axial modulus, $M$

UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

**Item 3**
Grain compressibility, $\gamma$

UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)
DEFAULT: 0

**Item 4**
Fraction, $f$, between 0 and 1

UNITS: dimensionless
DEFAULT: 0.5

**Item 5**
Exponent in the porosity-permeability relationship, $n$

UNITS: dimensionless
DEFAULT: 3

**Item 6 (i=1), Item 7 (i=2), ..., Item nc+5**
Langmuir curve parameter, $\beta_i$ for component $i$

UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)
DEFAULT: 0

**Item nc+6 ..., Item 2*nc +5**
Langmuir curve parameter, $\epsilon_i$ for component $i$.

UNITS: dimensionless
The Palmer-Mansoori model adjusts the pore volumes, \( V(P, Y) \), at the current pressure, \( P \) and gas phase composition \( Y = (y_1, y_2, \ldots, y_{nc}) \) using

\[
V(P, Y) = V_0 \left( 1 + \frac{c_m}{\phi_0} (P - P_0) + \frac{1}{\phi_0} \left( \frac{K}{M} - 1 \right) (E - E_0) \right)
\]

where

- \( \phi_0 \) is the initial porosity,
- \( V_0 \) is the pore volume at the initial pressure, \( P_0 \), and initial gas phase composition \( Y_0 \).

The term \( c_m \) is given as

\[
c_m = \frac{1}{M} - \left( \frac{K}{M} + f - 1 \right)
\]

The strain for component \( k \) is computed from the input Langmuir parameters by

\[
e_k = \frac{e_k \beta_k a_k P}{1 + \sum \beta_j a_j P}
\]

where \( a_k = \frac{L_k}{(\sum L_j)} \) represent the adsorbed mole fraction of component \( k \). These are computed assuming instant adsorption from the gas phase mole fractions \( Y \) and pressure \( P \) using the Langmuir storage capacities \( L_k(Y, P) \) (see keyword \textsc{langmext} or \textsc{langmuir}).

The total strain is computed as

\[
E = \Sigma e_k
\]

Following Palmer-Mansoori, assuming that permeability varies with porosity as

\[
k_k / k_0^n = \left( \frac{\phi}{\phi_0} \right)^n
\]

then the transmissibility multiplier is given by

\[
\left( 1 + \frac{c_m}{\phi_0} (P - P_0) + \frac{1}{\phi_0} \left( \frac{K}{M} - 1 \right) (E - E_0) \right)^n
\]

It is possible to output a table of the pore volume and permeability multipliers to the PRT file by using the keyword \textsc{rprkpm} in the \textsc{schedule} section.

The pore volume multiplier is limited, so the resulting pore volume cannot become less than \( 5 \times 10^{-6} \), unless this value is altered in \textsc{cbmopts} item 9, while the permeability multiplier is allowed to go to zero.

**Notes**

- The rock compaction model assumes that the input porosities are taken as the porosities at reservoir conditions. In areas with cells that are not connected to coal cells different rock compaction models can be used. Some care should be taken as the input porosities might be assumed to be given at a different reference pressure.
- The coal volume itself is kept constant, since usually this is large compared to the pore volumes. Changes due to compaction hence only marginally makes a difference on the overall composition of the adsorbed gas contents.
• For under saturated coal the desorption pressure and the equilibrium composition is used instead of the pressure in the pore space, thus honoring that the coal gas contents is not changing. Note also that the current model compute the strain as if the gas sorption is instant.

• **CBMOPTS** item 4 can be used to activate monitoring of passes through the rebound region (when the pore volume derivative changes sign). This enables the output of the number of passes through the rebound region using mnemonic **NPMREB** in **RPTSFED** or **RPRT**.

• Unless **CBMOPTS** item 6 is set to **NOPMSCAL**, the strain term (E) is scaled with the adsorption capacity multiplier.

**Example**

In **METRIC** units, and with **NCOMPS=2**, and **NTCREG=2** specified in keyword **REGDIMS**:

```
ROCKPAME
--K------M-----gcr--f--exp--beta1--beta2--eps1--eps2
3.03E04 3.99E04 0.0 0.0 3.0 0.023 0.002 0.0128 0.01 /--coalnum 1
2.50E04 3.00E04 0.0 0.0 3.0 0.026 0.003 0.01 0.0121 /--coalnum 2
/```

**Keywords**
ROCKPMAT  Thermal Palmer-Mansoori rock model data

This keyword is only available in a THERMAL run. It is used to specify data for the Thermal Palmer-Mansoori rock model ([Ref. 78] in ECLIPSE Technical Description).

The data comprises NTROCC (see keyword ROCKCOMP in the RUNSPEC section) rows of Thermal Palmer-Mansoori rock model data, each terminated by a slash (/). If a row is left blank, then its values are copied from the previous row.

Each row consists of 9 columns of data:

1. The bulk modulus, \( K \)
   
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

2. The constrained axial modulus, \( M \)
   
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

3. Grain compressibility, \( \gamma \)
   
   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)
   
   DEFAULT: 0

4. Langmuir curve parameter, \( \beta \)
   
   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)
   
   DEFAULT: 0

5. Langmuir curve parameter, \( c_l \)
   
   UNITS: dimensionless
   
   DEFAULT: 0

6. Fraction, \( f \), between 0 and 1
   
   UNITS: dimensionless
   
   DEFAULT: 0.5

7. Exponent in the porosity-permeability relationship, \( n \)
   
   UNITS: dimensionless
   
   DEFAULT: 3

8. Grain thermal expansivity, \( \alpha \)
   
   UNITS: 1/K (METRIC), 1/°R (FIELD), 1/K (LAB), 1/K (PVT-M)
   
   DEFAULT: 0.0

9. Grain thermal expansivity, \( c_v \)
   
   UNITS: kJ/m³/K (METRIC), Btu/ft³/°R (FIELD), J/cc/K (LAB), kJ/m³/K (PVT-M)
   
   DEFAULT: 0.0

The Thermal Palmer-Mansoori model adjusts the change of pore volumes, \( \Delta V(P, T) \), and total internal energy \( \Delta E^T(P, T) \), at the current pressure, \( P \), and temperature, \( T \), using
\[ \Delta V (P, T) = V (P_0, T_0) \left( \frac{c_m}{\varphi_0} \Delta P + \frac{1}{\varphi_0} \left( \frac{K}{M} - 1 \right) \alpha \Delta T \right) \]

\[ \Delta E^T = \Delta E^C \cdot \frac{c_v}{\alpha} \left( \frac{\varepsilon \beta P}{1 + \beta P} \right) \Delta P \]

where

- \( \varphi_0 \) is the initial porosity.
- \( V(P_0, T_0) \) is the pore volume at the initial pressure, \( P_0 \), and temperature, \( T \),
- \( \Delta E^C \) is the conventional change of the internal energy,
- \( \Delta E^T \) is the total change of the internal energy and
- \( c_m = \frac{1}{M} \left[ \frac{K}{M} + f - 1 \right] \) is the bulk elastic compressibility

This follows the conventional Palmer-Mansoori model, assuming that permeability varies with porosity as

\[ \frac{k}{k_0} = \left( \frac{\varphi}{\varphi_0} \right)^n \]

**Note:** The Thermal Palmer-Mansoori rock compaction model assumes that the input porosities are taken as the porosities at reservoir conditions.

**Example**

In FIELD units, and with NTROCC=3:

```
ROCKMAT
450000  600000  0  0.0016  0.0128 1* 1* 0.01 /
500000  700000  0  0.0010  0.0080 /
450000  600000 /
```
ROCKPROP  Property data for the base or cap rock

This keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section).

ROCKPROP specifies properties of the rock surrounding the reservoir. Rocks can be specified in any order, and in one or several keywords, but each rock should not be re-specified. The set of records must be terminated with a blank record containing just a slash (/).

Each record refers to a rock type and contains the following items of data:

1. Rock type identification number
   This should be a number between 1 and the maximum number of rocks, entered using ROCKDIMS in the RUNSPEC section.

2. Initial temperature
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

3. Rock conductivity
   UNITS: kJ/m/day/K (METRIC), Btu/ft/day/°R (FIELD), J/cm/hr/K (LAB), kJ/m/day/K (PVT-M)

4. Volumetric heat capacity of the rock
   UNITS: kJ/m³/K (METRIC), Btu/ft³/°R (FIELD), J/cc/K (LAB), kJ/m³/K (PVT-M)

5. Temperature-dependent part of the volumetric heat capacity of the rock
   UNITS: kJ/m³/K² (METRIC), Btu/ft³/°R² (FIELD), J/cc/K² (LAB), kJ/m³/K² (PVT-M)
   DEFAULT: 0.0.

6. Calculation method for heat loss
   This has the value ‘V’ for Vinsome and Westerveld or ‘N’ for numerical.
   DEFAULT: Vinsome.

7. Minimum temperature difference needed between block temperature and item 2 above to start the heat loss calculation in case of Vinsome and Westerveld model (‘V’ in item 6)
   The lower limit is 0; the upper limit is 10°C (18°F).
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
   DEFAULT: 0 (all unit sets)

The ROCKCON keyword is used to define which cells on the edge of the reservoir connect to each rock type (which reference to the global grid), and the ROCKCONL keyword is used to specify the local grid refinement connection data for these cap and base rocks.

See "Thermal option" in the ECLIPSE Technical Description for further information on this option.

Example

ROCKPROP
1 70 4 35.0 0.0 /
/
ROCKSPLV  Dual porosity rock volume partitioning for thermal calculations

If the dual porosity or dual permeability options and the thermal option are being used (by specifying the THERMAL or TEMP and DUALPORO or DUALPERM keywords in the RUNSPEC section), this keyword can be used to specify a multiplier which determines the proportion of total rock volume used in each porosity.

The rock volume partitioning using ROCKSPLV is calculated as follows. The set of cells belonging to a given spatial location (that is the same I and J, and all Ks differing by (grid height)/(number of porosities)) is defined to have a bulk volume which is the average bulk volume of the individual cells, and a pore volume which is the total of each of the cell pore volumes. The total rock volume \( R \) for the set is then the set bulk volume minus the set total pore volume. The ROCKSPLV values, \( r_i \) for each cell \( i \) in the set, are then used so that a rock volume of \( r_i R \) is assigned to cell \( i \).

It is not required to have the proportions sum to unity, so that a multiplier effect can also be achieved. But the total rock volume must first have been calculated to have a positive value.

If this keyword is specified, then this proportionality factor must be defined for all cells in the grid. There is no default value.

If this keyword is not specified, then the rock proportions will be calculated as described for the thermal option in its section on "Dual porosity" in the ECLIPSE Technical Description.

UNITS: dimensionless

Example

This defines all cell values for a grid of size 10 x 10 x 10 (5 layers in each of 2 porosities.) More rock is put into the matrix than into the fracture. For user-specified tuning purposes, only 90% of the rock volume is used, for example, for history matching.

```
ROCKSPLV
500*0.8 500*0.1 /
```
The data comprises NTROCC (see keyword ROCKCOMP in the RUNSPEC section) tables of rock compaction data, each terminated by a slash (/). Each table consists of at least 3 columns of data.

If the RKTRMDIR keyword has been specified, each table consists of 5 columns of data.

Apart from the last or first rows, values in the second and third columns may be defaulted, and such values are linearly interpolated.

All values in the fourth and fifth columns can be defaulted, and if this is done the values from the third column are used. Defaulting all these values is equivalent to not specifying RKTRMDIR.

1. The pressure.
   - Values should increase monotonically down the column, unless the 'STRESS' option has been used in the ROCKOPTS keyword, in which case the values should decrease monotonically down the column.
   - UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2. The corresponding pore volume multiplier.
   - Values should be level or increasing down the column.

3. The corresponding transmissibility multiplier.
   - Values should be level or increasing down the column. If RKTRMDIR has been specified, this column is the transmissibility multiplier in the x-direction.

4. If RKTRMDIR has been specified, this column is the transmissibility multiplier in the y-direction.

5. If RKTRMDIR has been specified, this column is the transmissibility multiplier in the z-direction.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see keyword TABDIMS). The NTROCC records correspond to the rock compaction table region numbers, which are set using ROCKNUM.

Note: In ECLIPSE 300, NTROCC can be set using the 13th item in the TABDIMS keyword; if the value is set in both places (TABDIMS and ROCKCOMP) the maximum is taken. See item 2 of keyword ROCKCOMP.

If the Rock Compaction is specified by the keyword ROCKCOMP in the RUNSPEC section then one of the following keywords: ROCKTAB, ROCKTABH or ROCK2D must be used.

The grid block pore volumes and transmissibilities are adjusted dynamically by multiplying the reference values (derived from the input data) by the corresponding factors in the table, interpolated at the instantaneous grid block pressures. If the OVERBURD keyword is present the cell overburden pressure is subtracted from the grid block pressure prior to interpolate the pore volume and transmissibility multipliers.

If the overburden pressure supplied in the OVERBURD keyword is greater than the fluid pressure (which it often will be), the effective fluid pressure ($P_{\text{eff, f}} = P_{\text{fluid}} - \text{Overburden}$) is negative. In this case the ROCKTAB table has to be supplied with negative pressures in column 1. The ROCKTAB table can be input in a more intuitive manner by using the 'STRESS' option in the ROCKOPTS keyword before the ROCKTAB keyword. In this case the pore volume and transmissibility multipliers are tabulated against the effective overburden pressure, $P_{\text{eff, ov}} = \text{Overburden} - P_{\text{fluid}}$, that are then positive.
If the rock compaction process is irreversible ('IRREVERS' specified in keyword ROCKCOMP), the table is interpolated at the minimum pressure reached in each grid block, to stop the pore space re-inflating if the pressure increases.

The transmissibility multipliers specified in the table are applied in a similar way to the static transmissibility multipliers entered using keywords MULTX, MULTY and MULTZ for example. The multiplier associated with the block (I, J, K) is applied to the transmissibility between the block and its positive-direction neighbors (I+1, J, K), (I, J+1, K), and (I, J, K+1). The transmissibility multipliers are also applied to the well connection transmissibility factors, and to non-neighbor connections.

In the current version of ECLIPSE 100, the transmissibility multiplier is obtained using the explicit value of the grid block pressure (or the explicit minimum pressure if the process is irreversible).

For ECLIPSE 100, coal bed methane option. The monotonicity restriction for the multipliers can be relaxed by setting item 89 of the OPTIONS keyword (coal bed methane runs only).

For ECLIPSE 300, Coal Bed Methane option. The monotonicity restriction for the multipliers can be relaxed by setting item 98 of the OPTIONS3 keyword (coal bed methane runs only).

See also keywords ROCKTABH, ROCK2D, ROCK2DTR, ROCKTABW, ROCKTSIG, OVERBURD and ROCKOPTS, and "Rock compaction" in the ECLIPSE Technical Description.

### Example

With NTROCC=2 and NPPVT\geq 4:

```
ROCKTAB
1000 .96 .98
2000 .99 .99
3000 1.0 1.0 /
4000 1.01 1.0 /
1000 .94 .97
2000 .98 .99
3000 1.0 1.0 /
4000 1.02 1.01 /
```

---

**Keywords**

1859
ROCKTABH  Hysteretic rock compaction data tables

The data comprises NTROCC (see keyword ROCKCOMP in the RUNSPEC section) tables of rock compaction data. A single table consists of at least 2 and up to NPPVT (see keyword TABDIMS in the RUNSPEC section) records, each terminated by a slash (/).

Each set of records gives curves of pore volume and transmissibility multipliers against pressure that define the elastic expansion and contraction of the rock. The elastic behavior is bounded at low pressure by the deflation curve, which is made up of the first points on each elastic curve. If the 'HYSTER' option is used (see below), the elastic curves are extrapolated to infinite pressure. If however the 'BOBERG' option is used, the elastic behavior is bounded at high pressure by the dilation curve, which is made up of the last points on each elastic curve. A table is terminated by a null record (that is a record with no data before the terminating slash).

![Hysteresis options](image)

*Figure 3.13. Hysteresis options*

Each record consists of 3 columns of data and a terminating slash. However, if the RKTRMDIR keyword has been specified, each table consists of 5 columns of data.

All values in the fourth and fifth columns can be defaulted and, if this is done, the values from the third column are used. Defaulting these values is equivalent to not specifying RKTRMDIR.

1. The pressure.
   Values should increase monotonically down the column, unless the 'STRESS' option has been used in the ROCKOPTS keyword, in which case the values should decrease monotonically down the column.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2. The corresponding pore volume multiplier.
   Values should be level or increasing down the column.

3. The corresponding transmissibility multiplier.
   Values should be level or increasing down the column.
   If RKTRMDIR has been specified, this column is the transmissibility multiplier in the x-direction.

4. If RKTRMDIR has been specified, this column is the transmissibility multiplier in the y-direction.

5. If RKTRMDIR has been specified, this column is the transmissibility multiplier in the z-direction.

There should be the same number of entries in each column of a given record. This number should not be less than 2 or greater than NPPVT (see keyword TABDIMS).
The NTROCC records correspond to the rock compaction table region numbers, which are set using ROCKNUM. Note that in ECLIPSE 300, NTROCC can be set using the 13th item in the TABDIMS keyword; if the value is set in both places the maximum is taken.

The ROCKTABH keyword must be used if the Hysteretic Rock Compaction option is selected ('HYSTER' or 'BOBERG' specified in keyword ROCKCOMP in the RUNSPEC section). The grid block pore volumes and transmissibilities are adjusted dynamically by multiplying the reference values (derived from the input data) by the corresponding factors in the table, interpolated at the instantaneous grid block pressures. If the OVERBURD keyword is present, the cell overburden pressure is subtracted from the grid block pressure prior to interpolating the pore volume and transmissibility multipliers.

**Note:** The curve to be used for both transmissibility and pore volume multipliers is determined by the pore volume multiplier data. The pore volume multiplier curves must therefore be distinct - it is not possible to have separate curves for the transmissibility multipliers but superposed curves for the pore volume multipliers.

If the overburden pressure supplied in the OVERBURD keyword is greater than the fluid pressure (which it often will be), the effective fluid pressure \( P_{\text{eff, fluid}} = P_{\text{fluid}} - \text{Overburden} \) is negative. In this case the ROCKTABH table has to be supplied with negative pressures in column 1. The ROCKTABH table can be input in a more intuitive manner by using the 'STRESS' option in the ROCKOPTS keyword before the ROCKTABH keyword. In this case the pore volume and transmissibility multipliers will be tabulated against the effective overburden pressure, \( P_{\text{eff, ov}} = \text{Overburden} - P_{\text{fluid}} \), which is then positive.

The transmissibility multipliers specified in the table are applied in a similar way to the static transmissibility multipliers entered using keywords MULTX, MULTY and MULTZ for example. The multiplier associated with the block (I, J, K) is applied to the transmissibility between the block and its positive-direction neighbors (I+1, J, K), (I, J+1, K), and (I, J, K+1). The transmissibility multipliers are also applied to the well connection transmissibility factors, and to non-neighbor connections.

In the current version of ECLIPSE 100, the transmissibility multiplier is obtained using the explicit value of the grid block pressure.

There is a potential problem when using transmissibility multipliers with the BOBERG hysteresis option. When the pressure exceeds the elastic limits, pore volume and transmissibility multipliers are taken either from the deflation curves or dilation curves. The elastic limits are determined by finding where the pore volume elastic curves intersect the pore volume deflation and dilation curves. Therefore, the transmissibility is discontinuous at the elastic limits if the transmissibility multipliers differ significantly from the pore volume multipliers. To ensure this problem does not arise, the transmissibility multipliers can be set equal to the pore volume multipliers, or not used, by setting them equal to 1.

See also keywords ROCKTAB, ROCK2D, ROCK2DTR, ROCKTABW, ROCKTHSG, OVERBURD and ROCKOPTS, and "Rock compaction" in the ECLIPSE Technical Description.

It is possible to define a minimum and maximum pore volume multiplier, for each table, in the BOBERG model using the keyword RKPVMNMX. Given the pressure, if the calculated pore volume multiplier is less than the minimum value set using the RKPVMNMX keyword, then the multipliers are redefined on the elastic curve. Similarly, if the calculated multiplier is greater than the maximum value set using the RKPVMNMX keyword, then the multipliers are redefined on the elastic curve.

**Example**

With NTROCC=2 and NPPVT≥4:
In this example the deflation curve for Table 1 is:

<table>
<thead>
<tr>
<th>Value</th>
<th>Table 1</th>
<th>Table 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>1000</td>
<td>0.96</td>
<td>0.94</td>
</tr>
<tr>
<td>2000</td>
<td>0.99</td>
<td>0.98</td>
</tr>
<tr>
<td>3000</td>
<td>1.00</td>
<td>0.99</td>
</tr>
<tr>
<td>4000</td>
<td>1.01</td>
<td>1.02</td>
</tr>
</tbody>
</table>

and the dilation curve (if used) is:

<table>
<thead>
<tr>
<th>Value</th>
<th>Table 1</th>
<th>Table 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>4000</td>
<td>0.99</td>
<td>0.995</td>
</tr>
<tr>
<td>4500</td>
<td>0.998</td>
<td>0.998</td>
</tr>
<tr>
<td>5000</td>
<td>1.01</td>
<td>1.01</td>
</tr>
<tr>
<td>5500</td>
<td>1.02</td>
<td>1.02</td>
</tr>
</tbody>
</table>
ROCKTABW  Water induced compaction tables

The data comprises NTROCC (see keyword ROCKCOMP in the RUNSPEC section) tables of water induced compaction data, each terminated by a slash (/).

Each table consists of 3 columns of data:

1. The water saturation increase from initial value.
   Values should increase monotonically down the column.
2. The corresponding pore volume multiplier.
   Values should be level or decreasing down the column.
3. The corresponding transmissibility multiplier.

There must be the same number of entries in each column of a given table. The number should not be less than 2 or greater than NSSFUN (see keyword TABDIMS in the RUNSPEC section).

The NTROCC records correspond to the rock compaction table region numbers, which are set using ROCKNUM.

The ROCKTABW keyword should be used in conjunction with either the ROCK, ROCKTAB or ROCKTABH keywords which specify the compaction as a function of pressure. The overall multipliers applied to the pore volume, and the transmissibility will be the product of the pressure induced compaction multipliers and the water induced compaction multipliers. Alternatively the overall compaction multipliers can be entered as a two-dimensional table in pressure and water saturation using the ROCKWNOD, ROCK2D and ROCK2DTR keywords.

The water saturation used when looking up the table is the maximum attained water saturation for a grid block minus the water saturation at the start of the run. Thus the water induced compaction is assumed to be irreversible and is tabulated relative to the change in water saturation rather than the absolute saturation. See "Rock compaction" in the ECLIPSE Technical Description for details.

See also keywords ROCKTAB, ROCKTABH, ROCK2D, ROCK2DTR and ROCKWNOD.

Example

With NTROCC=2 and NSSFUN≥4:

```
ROCKTABW
  0.0  1.0  1.0
  0.4  0.93 0.98
  1.0  0.93 0.96 /
  0.0  1.0  1.0
  0.3  1.0  1.0
  0.4  0.90 0.98
  1.0  0.90 0.96 /
```

Keywords

```plaintext
ECLIPSE Reference Manual
ROCKTABW
Water induced compaction tables

The data comprises NTROCC (see keyword ROCKCOMP in the RUNSPEC section) tables of water induced compaction data, each terminated by a slash (/).

Each table consists of 3 columns of data:

1. The water saturation increase from initial value.
   Values should increase monotonically down the column.
2. The corresponding pore volume multiplier.
   Values should be level or decreasing down the column.
3. The corresponding transmissibility multiplier.

There must be the same number of entries in each column of a given table. The number should not be less than 2 or greater than NSSFUN (see keyword TABDIMS in the RUNSPEC section).

The NTROCC records correspond to the rock compaction table region numbers, which are set using ROCKNUM.

The ROCKTABW keyword should be used in conjunction with either the ROCK, ROCKTAB or ROCKTABH keywords which specify the compaction as a function of pressure. The overall multipliers applied to the pore volume, and the transmissibility will be the product of the pressure induced compaction multipliers and the water induced compaction multipliers. Alternatively the overall compaction multipliers can be entered as a two-dimensional table in pressure and water saturation using the ROCKWNOD, ROCK2D and ROCK2DTR keywords.

The water saturation used when looking up the table is the maximum attained water saturation for a grid block minus the water saturation at the start of the run. Thus the water induced compaction is assumed to be irreversible and is tabulated relative to the change in water saturation rather than the absolute saturation. See "Rock compaction" in the ECLIPSE Technical Description for details.

See also keywords ROCKTAB, ROCKTABH, ROCK2D, ROCK2DTR and ROCKWNOD.

Example

With NTROCC=2 and NSSFUN≥4:

```
ROCKTABW
  0.0  1.0  1.0
  0.4  0.93 0.98
  1.0  0.93 0.96 /
  0.0  1.0  1.0
  0.3  1.0  1.0
  0.4  0.90 0.98
  1.0  0.90 0.96 /
```

Keywords

1863
ROCKTHSG  Hysteretic rock compaction data tables in dual porosity runs

In addition to the ROCKTABH tables, dual porosity models can also contain pressure tables of the sigma multiplier modifying the matrix-fracture coupling as a result of rock compaction.

The data comprises NTROCC (see item 2 of keyword ROCKCOMP in the RUNSPEC section) tables of rock compaction data. A single table consists of at least 2 and up to NPPVT (see keyword TABDIMS in the RUNSPEC section) records, each terminated by a slash (/).

Each set of records gives curves of sigma multipliers against pressure that define the elastic expansion and contraction of the dual porosity rock. The elastic behavior is bounded at low pressure by the deflation curve, which is made up of the first points on each elastic curve. If the 'HYSTER' option is used (see below), the elastic curves are extrapolated to infinite pressure. If, however, the 'BOBERG' option is used, the elastic behavior is bounded at high pressure by the dilation curve, which is made up of the last points on each elastic curve. Each table is terminated by a null record (that is a record with no data before the terminating slash).

![Figure 3.14. Hysteresis options](image)

Each record consists of 2 columns of data and a terminating slash.

1. The pressure.
   - Values should increase monotonically down the column, unless the 'STRESS' option has been used in the ROCKOPTS keyword, in which case the values should decrease monotonically down the column.
   - **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2. The corresponding sigma multiplier.
   - Values should be level or increasing down the column.

There must be the same number of entries in each column of a given record. This number should not be less than 2 or greater than NPPVT (see keyword TABDIMS). The NTROCC records correspond to the rock compaction table region numbers, which are set using ROCKNUM.

**Note:** NTROCC can be set using the 13th item of the TABDIMS keyword; if the value is set in both places (TABDIMS and ROCKCOMP) the maximum is taken.
Note: Although ROCKNUM regions always cover the entire grid (even when defaulted), only the ROCKNUM numbers of the matrix cells are used to obtain the ROCKTHSG tables for the matrix-fracture coupling modification. ROCKNUM for fracture cells is ignored.

The ROCKTHSG keyword may be used only if the Hysteretic Rock Compaction option is selected (‘HYSTER’ or ‘BOBERG’ specified in keyword ROCKCOMP in the RUNSPEC section). The grid-block matrix-fracture coupling transmissibilities are adjusted dynamically by multiplying the reference values (derived from the input data) by the corresponding factors in the table, interpolated at the instantaneous grid block pressures. If the OVERBURD keyword is present, the cell overburden pressure is subtracted from the grid block pressure prior to interpolating the sigma multipliers.

If the overburden pressure supplied in the OVERBURD keyword is greater than the fluid pressure (which it often will be), the effective fluid pressure \( P_{\text{eff, f}} = P_{\text{fluid}} - \text{Overburden} \) is negative. In this case the ROCKTHSG table has to be supplied with negative pressures in column 1. The ROCKTHSG table can be input in a more intuitive manner by using the ‘STRESS’ option in the ROCKOPTS keyword before the ROCKTHSG keyword. In this case the sigma multipliers will be tabulated against the effective overburden pressure, \( P_{\text{eff, ov}} = \text{Overburden} - P_{\text{fluid}} \), which is then positive.

Note: The implicit value of the grid block pressure is used for the calculations, as it is for ECLIPSE 300. This is different from the treatment of the transmissibility multipliers in the ROCKTABH keyword in ECLIPSE 100, where the pressure is explicit.

Either DUALPORO or DUALPERM keyword must be specified in the RUNSPEC section, and valid ROCKTABH data must be supplied, in order to enable ROCKTHSG input. The deflation pressures, as well as the dilation pressures in the BOBERG case, must be identical between the corresponding tables of ROCKTABH and ROCKTHSG. This is to ensure the same hysteretic behavior in both sets of tables. Note that the elastic curves may differ between the two sets.

See also keywords ROCKTSIG, RKPVMINMX, OVERBURD and ROCKOPTS, and "Rock compaction" in the ECLIPSE Technical Description.

Example

With NTROCC=2 specified keyword ROCKCOMP and NPPVT\geq4 specified in keyword TABDIMS:

```
ROCKTHSG
-- Table 1
  1000  .98
  2000  .985
  3000  .99
  4000  .995 /
  2000  .99
  3000  .995
  4500  .998 /
  3000  1.0
  5000  1.01 /
  4000  1.01
  5500  1.02 /
-- Table 2
  1000  .97
  2700  .97 /
  2000  .99
  3500  .99 /
  3000  1.0
  4300  1.0 /
```
In this example the deflation curve for Table 1 is:

<table>
<thead>
<tr>
<th>Value</th>
<th>Coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>1000</td>
<td>0.98</td>
</tr>
<tr>
<td>2000</td>
<td>0.99</td>
</tr>
<tr>
<td>3000</td>
<td>1.00</td>
</tr>
<tr>
<td>4000</td>
<td>1.01</td>
</tr>
</tbody>
</table>

and the dilation curve (if used) is:

<table>
<thead>
<tr>
<th>Value</th>
<th>Coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>4000</td>
<td>0.995</td>
</tr>
<tr>
<td>4500</td>
<td>0.998</td>
</tr>
<tr>
<td>5000</td>
<td>1.01</td>
</tr>
<tr>
<td>5500</td>
<td>1.02</td>
</tr>
</tbody>
</table>
ROCKTRMX Transmissibility variation parameters in X-direction

This keyword allows you to input transmissibility variation parameters for each grid cell in the X-direction. Currently, ECLIPSE 300 allows the transmissibility to vary along with pore volume as a function of pressure (for instance, ROCKTAB and ROCKTABH keyword input). This approach is quite simplified as it does not take into account various important aspects:

- The transmissibility value in a cell which has a high porosity value would change differently to a cell which has a low porosity value.
- Some cells, around the well for instance, may require different modifications to be applied to them for history matching.

The information provided with this keyword will be used if you set ROCKCOMP keyword item 4 to EXP, which will define the transmissibility as an exponential function of porosity, given by the following expression:

\[ \frac{\kappa}{\kappa_0} = e^{\kappa_{\text{mult}} \left( \frac{\phi - \phi_0}{1 - \phi_0} \right)} \]

where

- \(\kappa_0\) is the initial transmissibility,
- \(\phi_0\) is the initial porosity and
- \(\kappa_{\text{mult}}\) is the user-defined multiplier factor defined by the ROCKTRMX keyword.

Any transmissibility multiplier previously set using the ROCKTAB or ROCKTABH keywords will be ignored if the ROCKTRMX keyword is defined.

The transmissibility multipliers are also applied to the well connection factors and to non-neighbor connections.

This keyword is not compatible with the NINEPOIN and MPFA keywords.

Values entered using this keyword should range from 0 to 1E10. The default value will be set to 0 which corresponds to no transmissibility variation with fluid porosity.

The keyword should be followed by one real number for every grid block in the current input box, specifying the required water saturation. The data field should be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest (from 1 to \(\text{NDIVIX}\)), followed by the Y-axis index (from 1 to \(\text{NDIVIY}\)), and finally the Z-axis index (from 1 to \(\text{NDIVIZ}\)). \(\text{NDIVIX}\), \(\text{NDIVIY}\) and \(\text{NDIVIZ}\) are specified in RUNSPEC keyword DIMENS.

Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*).

Example

With \(\text{NDIVIX}=20\), \(\text{NDIVIY}=6\) and \(\text{NDIVIZ}=5\) specified in RUNSPEC keyword DIMENS:
<table>
<thead>
<tr>
<th>ROCKTRMX</th>
</tr>
</thead>
<tbody>
<tr>
<td>120*0</td>
</tr>
<tr>
<td>120*0.1</td>
</tr>
<tr>
<td>120*0.5</td>
</tr>
<tr>
<td>120*0.7</td>
</tr>
<tr>
<td>120*1</td>
</tr>
</tbody>
</table>
ROCKTRMY Transmissibility variation parameters in Y-direction

This keyword allows you to input transmissibility variation parameters for each grid cell in the Y-direction. Currently, ECLIPSE 300 allows the transmissibility to vary along with pore volume as a function of pressure (for instance, ROCKTAB and ROCKTABH keyword input). This approach is quite simplified as it does not take into account various important aspects:

• The transmissibility value in a cell which has a high porosity value would change differently to a cell which has a low porosity value.
• Some cells, around the well for instance, may require different modifications to be applied to them for history matching.

The information provided with this keyword will be used if you set ROCKCOMP keyword item 4 to EXP, which will define the transmissibility as an exponential function of porosity, given by the following expression:

\[
\frac{\kappa}{\kappa_0} = e^{\kappa_{\text{mult}}\left(\frac{\phi - \phi_0}{1 - \phi_0}\right)}
\]

where

\(\kappa_0\) is the initial transmissibility,

\(\phi_0\) is the initial porosity and

\(\kappa_{\text{mult}}\) is the user-defined multiplier

Any transmissibility multiplier previously set using the ROCKTAB or ROCKTABH keywords will be ignored if the ROCKTRMY keyword is defined.

The transmissibility multipliers are also applied to the well connection factors and to non-neighbor connections.

This keyword is not compatible with the NINEPOIN and MPFA keywords.

Values entered using this keyword should range from 0 to 1E10. The default value will be set to 0 which corresponds to no transmissibility variation with fluid porosity.

The keyword should be followed by one real number for every grid block in the current input box, specifying the required water saturation. The data field should be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest (from 1 to NDIVIX), followed by the Y-axis index (from 1 to NDIVIY), and finally the Z-axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS.

Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*).

Example

With NDIVIX=20, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS:
ROCKTRMY
120*0
120*0.1
120*0.5
120*0.7
120*1
/
ROCKTRMZ  Transmissibility variation parameters in Z-direction

This keyword allows you to input transmissibility variation parameters for each grid cell in the Z-direction. Currently, ECLIPSE 300 allows the transmissibility to vary along with pore volume as a function of pressure (for instance, ROCKTAB and ROCKTABH keyword input). This approach is quite simplified as it does not take into account various important aspects:

- The transmissibility value in a cell which has a high porosity value would change differently to a cell which has a low porosity value.
- Some cells, around the well for instance, may require different modifications to be applied to them for history matching.

The information provided with this keyword will be used if you set ROCKCOMP keyword item 4 to EXP, which will define the transmissibility as an exponential function of porosity, given by the following expression:

\[
\frac{\kappa}{\kappa_0} = e^{\kappa_{\text{mult}} \left( \frac{\phi - \phi_0}{1 - \phi_0} \right)}
\]

where

- \(\kappa_0\) is the initial transmissibility,
- \(\phi_0\) is the initial porosity and
- \(\kappa_{\text{mult}}\) is the user-defined multiplier factor defined by the ROCKTRMZ keyword.

Any transmissibility multiplier previously set using the ROCKTAB or ROCKTABH keywords will be ignored if the ROCKTRMZ keyword is defined.

The transmissibility multipliers are also applied to the well connection factors and to non-neighbor connections.

This keyword is not compatible with the NINEPOIN and MPFA keywords.

Values entered using this keyword should range from 0 to 1E10. The default value will be set to 0 which corresponds to no transmissibility variation with fluid porosity.

The keyword should be followed by one real number for every grid block in the current input box, specifying the required water saturation. The data field should be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest (from 1 to NDIVIX), followed by the Y-axis index (from 1 to NDIVIY), and finally the Z-axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS.

Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*).

Example

With NDIVIX=20, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS:
ROCKTRMZ
120*0
120*0.1
120*0.5
120*0.7
120*1   /
ROCKTSIG  Rock compaction data tables in dual porosity runs

In addition to the ROCKTAB tables, dual porosity models can also contain pressure tables of the sigma multiplier modifying the matrix-fracture coupling as a result of rock compaction.

The data comprises NTROCC (see item 2 of keyword ROCKCOMP in the RUNSPEC section) tables of Rock Compaction data, each terminated by a slash (/).

Each table consists of 2 columns of data:

1. The pressure.
   - Values should increase monotonically down the column, unless the 'STRESS' option has been used in the ROCKOPTS keyword, in which case the values should decrease monotonically down the column.
   - UNITS: barsa (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)

2. The corresponding sigma multiplier.
   - Values should be level or increasing down the column.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see keyword TABDIMS). The NTROCC records correspond to the rock compaction table region numbers, which are set using ROCKNUM.

Note: NTROCC can be set using the 13th item of the TABDIMS keyword; if the value is set in both places (TABDIMS and ROCKCOMP) the maximum is taken.

Note: Although ROCKNUM regions always cover the entire grid (even when defaulted), only the ROCKNUM numbers of the matrix cells are used to obtain the ROCKTSIG tables for the matrix-fracture coupling modification. ROCKNUM for fracture cells is ignored.

The ROCKTSIG keyword may be used only if the reversible or irreversible rock compaction option is selected ('REVERS' or 'IRREVERS' specified in keyword ROCKCOMP in the RUNSPEC section). The grid block matrix-fracture coupling transmissibilities are adjusted dynamically by multiplying the reference values (derived from the input data) by the corresponding factors in the table, interpolated at the instantaneous grid block pressures. If the OVERBURD keyword is present, the cell overburden pressure is subtracted from the grid block pressure prior to interpolating the sigma multipliers.

If the overburden pressure supplied in the OVERBURD keyword is greater than the fluid pressure (which it often will be), the effective fluid pressure \( P_{\text{eff fl}} = P_{\text{fluid}} - \text{Overburden} \) is negative. In this case the ROCKTSIG table has to be supplied with negative pressures in column 1. The ROCKTSIG table can be input in a more intuitive manner by using the 'STRESS' option in the ROCKOPTS keyword before the ROCKTSIG keyword. In this case the sigma multipliers will be tabulated against the effective overburden pressure, \( P_{\text{eff ov}} = \text{Overburden} - P_{\text{fluid}} \), which is then positive.

If the rock compaction process is irreversible ('IRREVERS' specified in keyword ROCKCOMP), the table is interpolated at the minimum pressure reached in each grid block, in accordance with the ROCKTAB behavior which stops the pore space re-inflating if the pressure increases.
**Note:** The implicit value of the grid block pressure is used for the calculations, as it is for ECLIPSE 300. This is different from the treatment of the transmissibility multipliers in the ROCKTAB keyword in ECLIPSE 100, where the pressure is explicit.

Either DUALPORO or DUALPERM keyword must be specified in the RUNSPEC section, and valid ROCKTAB data must be supplied, in order to enable ROCKTSIG input.

See also keywords ROCKTHSG, OVERBURD and ROCKOPTS, and "Rock compaction" in the ECLIPSE Technical Description.

**Example**

With NTROCC=2 specified in keyword ROCKCOMP and NPPVT ≥4 specified in keyword TABDIMS:

```
ROCKSIG
 1000  .98
 2000  .99
 3000  1.0
 4400  1.0  /
 1000  .97
 2000  .99
 3000  1.0
 4000  1.01  /
```
ROCKV

Grid block rock volumes

This keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section).

ROCKV should be followed by one real number for every grid block in the current input box, specifying the grid block rock volumes.

This keyword is used to overwrite the rock volumes calculated from the GRID data. Setting a zero rock volume causes a cell to be inactive if the cell also has a zero pore volume.

UNITs: \( \text{rm}^3 \) (\text{METRIC}), \( \text{rb} \) (\text{FIELD}), \( \text{rcc} \) (\text{LAB}), \( \text{rm}^3 \) (\text{PVT-M})

Example

```
ROCKV
24*1.324E4 24*0 12*1876 /
```
ROCKWNOD  The water saturation node values for water-induced compaction tables

The data comprises NTROCC (see keyword ROCKCOMP in the RUNSPEC section) tables of water saturation values, each terminated by a slash (/). The tables define the water saturation node values for the 2-D water induced compaction tables entered with keywords ROCK2D and ROCK2DTR.

Each table consists of up to NSSFUN (see keyword TABDIMS in the RUNSPEC section) values of the water saturation increase, that is the current water saturation minus its initial value. Within each table the values should be positive and strictly increasing. The number of entries in each table must correspond to the number of pore volume and transmissibility multiplier entries in the corresponding ROCK2D and ROCK2DTR tables.

The NTROCC tables correspond to the rock compaction table region numbers, which are set using keyword ROCKNUM.

The water saturation used when looking up the tables is the maximum attained water saturation for a grid block minus the water saturation at the start of the run. Thus the water induced compaction is assumed to be irreversible and is tabulated relative to the change in water saturation rather than the absolute saturation. See "Rock compaction" in the ECLIPSE Technical Description for details.

See also keywords ROCK2D, ROCK2DTR and ROCKTABW.

Example

With NTROCC=2 and NPPVT≥5:

```
ROCKWNOD
  0.0
  0.3
  0.4
  0.7
  1.0  /
  0.0
  0.4
  0.7
  1.0  /

ROCK2D
  0.0  0.88
    0.88
    0.88
    0.88
    0.88 /
  1000.0  0.92
      0.92
      0.91
      0.90
      0.90 /
  9000.0  1.0
    1.0
    0.98
    0.94
    0.94 /
 /
  0.0  0.88
    0.88
    0.88
    0.88
    0.88 /
  1000.0  0.92
      0.91
      0.90
      0.90
      0.90 /
  9000.0  1.0
```

Keywords

1876
Specifies cell residual oil composition

This keyword specifies the initial residual oil composition values for each cell. It can only be used with the SOR option. When flashed at the initial cell pressure the fluid must be a single phase oil.

The keyword should be followed by $N_x \cdot N_y \cdot N_z \cdot N_c$ values in normal order, with the X index changing fastest.

Example

```
ROMF
300*0.00
300*0.00
300*1.00
/
```
**RPRKPAME  Output ROCKPAME table for a cell**

This keyword can be used to output a table of pore volume and permeability multipliers versus pressure to the PRT file. The keyword is intended for validation of the input data in connection with the use of the ROCKPAME rock compaction model. A table is generated when the keyword is read in the SCHEDULE section. The keywords have the following items and are ended with a slash.

1. **I** - location of grid block  
   No default
2. **J** - location of grid block  
   No default
3. **K** - location of grid block  
   No default
4. **Lower pressure of table**  
   **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)  
   **DEFAULT:** 1.0 barsa
5. **Upper pressure of table**  
   **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)  
   **DEFAULT:** 600.0 barsa
6. **Number of pressure values in table**  
   **DEFAULT:** 20

The keyword can only be applied to global grid cells (not LGRs) and cannot be used in parallel mode.

**Note:** When using the time-dependent sorption model, a coal region number, **COALNUM**, needs to be defined for the connecting dual/multi-porosity grid cell. When using the instant sorption model, a coal region number needs to be defined for the given grid cell.

**Example**

```
RPRKPAME 1 1 2 100 200 50/
```

---

**Keywords**
Controls on output from GRID section

The keyword should be followed by a string of mnemonics to request output of grid data to the print file. The appropriate keywords used to input grid data, such as DX and DY, may be used as mnemonics. Aliases may be used; for example DR may be used in place of DX and DTHETA may be used in place of DY. The data should be terminated by a slash (/).

Note the following for ECLIPSE 100:

- Older ECLIPSE 100 data sets may use integer controls, not mnemonics. The meaning of these is documented in "RPTGRID" in the ECLIPSE Technical Description.
- Entering the mnemonic NOTHING clears all the reporting flags.
- For additional control over the output, integer values greater than one may be assigned to selected mnemonics using the syntax:
  
  mnemonic=integer

  where there must be no spaces either side of the equals sign.

Output requested using the keyword RPTGRID applies only to the global grid. To request output for local grids, use RPTGRIDL.

The following table lists the mnemonics available. Mnemonics available are identified by an x in the appropriate column.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALLNNC</td>
<td>Output of all global-grid non-neighbor connections from all sources</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>AQUCON</td>
<td>Output of numerical aquifer connections</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>AQUNUM</td>
<td>Output of numerical aquifer definitions</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>COALNUM</td>
<td>Output of coal region numbers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>COALV</td>
<td>Output of grid block coal volumes (Coal Bed Methane)</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>COLLAPSE</td>
<td>Output of VE compression flag</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>COORD</td>
<td>Output of coordinate lines</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>COORDSYS</td>
<td>Output of coordinate systems</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>DEPTH</td>
<td>Output of grid block center depths</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DIFFMMF</td>
<td>Output of Matrix/Fracture diffusivity multipliers</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>DIFFMX</td>
<td>Output of X diffusivity multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DIFFMY</td>
<td>Output of Y diffusivity multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DIFFMZ</td>
<td>Output of Z diffusivity multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DIFFX or DIFFTX</td>
<td>Output of X diffusivities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DIFFY or DIFFTY</td>
<td>Output of Y diffusivities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DIFFZ or DIFFTZ</td>
<td>Output of Z diffusivities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>---------------</td>
<td>----------------------------------------------------------------------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>DOMAINS</td>
<td>Output of domains (Parallel Option). See also the DOMAINS keyword. The output consists of: • The solver direction, which ECLIPSE calculates automatically but which may be modified by using the SOLVDIRS keyword. • The split of the outer solver dimension into domains. The split can be entered using the DOMAINS keyword, otherwise ECLIPSE will calculate it automatically.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>DPCON or SIGMA</td>
<td>Output of dual porosity connections</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>DNUM</td>
<td>Output of dual porosity/single porosity regions</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DX</td>
<td>Output of X direction grid block sizes</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DY</td>
<td>Output of Y direction grid block sizes</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DZ</td>
<td>Output of Z direction grid block sizes</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DZMTRXV</td>
<td>Output of $D_{zmtrx}$ values</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DZNET</td>
<td>Output of net Z-direction grid block sizes</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>EXTHOST or</td>
<td>Output the host cell map for Pebi local grid refinements supplied using the EXFIN keyword</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>EXTFIN</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FAULTS</td>
<td>Output of fault data</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>FLUXNUM</td>
<td>Output of flux region numbers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>HEATTX</td>
<td>X thermal transmissibility</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>HEATTY</td>
<td>Y thermal transmissibility</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>HEATTZ</td>
<td>Z thermal transmissibility</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>HM* ***</td>
<td>Outputs multipliers for gradient parameters</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>IOCN</td>
<td>Output of active cell numbers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>IONROCK</td>
<td>Rock ion exchange capacity</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ISOLNUM</td>
<td>Output of independent reservoir region numbers</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>JFUNCR</td>
<td>Output of Leverett J-Function data specified using JFUNCR keyword.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>KOVERD</td>
<td>Output of K/D values (Surfactant Model)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>LIMITS</td>
<td>Outputs limits (max/min) of grid arrays</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>LX</td>
<td>Output of matrix block size, X-direction (Viscous Displacement Option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>LY</td>
<td>Output of matrix block size Y-direction</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>LZ</td>
<td>Output of matrix block size Z-direction</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MIDS</td>
<td>Output of cell mid-point depths</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MINPV or</td>
<td>Output of cells with pore volume less than the input minimum values</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MINPVV</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>--------------</td>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>MPFANUM</td>
<td>Output the <strong>MPFANUM</strong> data when using the multi-point flux approximation</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MFPFORTH</td>
<td>Output the multi-point flux k-orthogonality error</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MPFTRAN</td>
<td>Output the multi-point flux forward transmissibilities</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MPFTRREV</td>
<td>Output the multi-point flux reverse transmissibilities</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MULTNUM</td>
<td>Output of regions for inter-region transmissibility multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MULTPV</td>
<td>Output of the pore volume multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MULTREGP</td>
<td>Output region pore volume multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MULTREGT</td>
<td>Output of multipliers between regions</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MULTX</td>
<td>Output of X transmissibility multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MULTY</td>
<td>Output of Y transmissibility multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MULTZ</td>
<td>Output of Z transmissibility multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>NINENUM</td>
<td>Output the <strong>NINENUM</strong> data when using the nine-point scheme</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>NNC</td>
<td>Output of non-neighbor connections</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>NTG</td>
<td>Output of grid block net-to-gross ratios</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>OPERNUM</td>
<td>Output of regions for use with <strong>OPERATER</strong> keyword</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PERMX</td>
<td>Output of X direction permeabilities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PERMY</td>
<td>Output of Y direction permeabilities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PERMZ</td>
<td>Output of Z direction permeabilities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PERMXY, PERMYZ or PERMZX</td>
<td>Output of all tensor permeability coefficients</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PINCH or PINCHREG</td>
<td>Output of pinchout data</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PINCHNUM</td>
<td>Output of pinchout region numbers</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>POLYMER or FOAM</td>
<td>Output of 1/(porosity*area) (Polymer and Foam Models)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PORO</td>
<td>Output of grid block porosities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PORV</td>
<td>Output of grid block pore volumes</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PORVF</td>
<td>Output of grid block fluid pore volumes (Coal Bed Methane option)</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>ROCKFRAC</td>
<td>Output of grid block rock volume fractions (Coal Bed Methane option)</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>ROCKVOL</td>
<td>Output of bulk (net and gross) rock volumes to the print file. In addition, if <strong>FIPNUM</strong> regions are defined, the bulk rock volumes for each region will be output.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Sigmagdv</td>
<td>Output of <strong>Sigmagd</strong> values (gravity drainage option)</td>
<td>x</td>
<td></td>
</tr>
</tbody>
</table>
### Table 3.29: RPTGRID output controls

The coordinate line data (COORD) and the corner depth data (ZCORN) can be written out to the DEBUG file in keyword form, by setting the COORD and ZCORN controls greater than 1. This keyword data can be read directly into the program. When used in conjunction with the BOUNDARY keyword, a grid corresponding to a small portion of the field can easily be generated. Note that block centered grids input using the DX, DY, DZ, TOPS keywords can also be output in corner point form.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>SIGMAK</td>
<td>Output of the processed ( \frac{f_x \cdot K_x}{l_x^2} + \frac{f_y \cdot K_y}{l_y^2} + \frac{f_z \cdot K_z}{l_z^2} ) term for the matrix-fracture transmissibility in dual porosity runs, if item 5 of the LTOSIGMA keyword is set to ALL.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SIGMAV</td>
<td>Output of Sigma values</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOLVMUM</td>
<td>Output the mapping between cells in the input order and the order used for the nested factorization solver when using PEBI grid data and supplied by the SOLVMUM keyword.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>THCAVE</td>
<td>Output of average thermal conductivities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>THCGAS</td>
<td>Output of gas phase thermal conductivities</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>THCOIL</td>
<td>Output of oil phase thermal conductivities</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>THCONR</td>
<td>Output of thermal rock and fluid conductivities</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>THCROCK</td>
<td>Output of rock thermal conductivities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>THCSOLID</td>
<td>Output of solid phase thermal conductivities</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>THCWATER</td>
<td>Output of water phase thermal conductivities</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>TOPS</td>
<td>Output of grid block top depths</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TRANX</td>
<td>Output of X transmissibilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TRANY</td>
<td>Output of Y transmissibilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TRANZ</td>
<td>Output of Z transmissibilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TRTHERM</td>
<td>Output of thermal transmissibilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>VELFLO</td>
<td>Output of (1/(\text{porosity} \times \text{area})) Only allowed if velocity or flow output requested via summary mnemonics or restart output</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ZCORN</td>
<td>Output of cell corner depths</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Control</td>
<td>Setting</td>
<td>Definition</td>
<td></td>
</tr>
<tr>
<td>---------</td>
<td>---------</td>
<td>------------</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>ZCORN keyword written to the DEBUG file in floating point format (F12.2)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>ZCORN keyword written in exponent format (1PE12.6)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Table 3.30: RPTGRID COORD and ZCORN control**

ECLIPSE 100 Grid-valued output values in certain types of cell have the decimal point replaced with a special character for ease of identification.

<table>
<thead>
<tr>
<th>Character replacing the decimal point</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td>Coarsened cell</td>
</tr>
<tr>
<td>T</td>
<td>Compressed vertical equilibrium — cell is the top (representative) cell in a column of merged cells</td>
</tr>
<tr>
<td>V</td>
<td>Compressed vertical equilibrium — cell is in a column of merged cells but not the top (representative) cell</td>
</tr>
<tr>
<td>R</td>
<td>Cell is refined (contains an LGR)</td>
</tr>
<tr>
<td>F</td>
<td>In a USEFLUX run using pressure/saturation boundary conditions, flow is specified in this cell</td>
</tr>
<tr>
<td>*</td>
<td>Cell contains a well connection</td>
</tr>
</tbody>
</table>

**Table 3.31: Output format conventions**

**Examples**

**Example 1**

```
RPTGRID
COORD TRANX TRANY TRANZ ALLNCC /
```

**Example 2**

Changing the level of output

```
RPTGRID
COORD=2 ZCORN=2 /
```
RPTGRIDL  

GRID section data output for local refined grids

RPTGRIDL is used to request output of GRID section data for local grids. The keyword data is the same as for the first 57 mnemonics described in the RPTGRID keyword, except for the following switches (see also section "RPTGRID - definition with integer controls" in the ECLIPSE Technical Description):

**Control/Mnemonic**

24  **ALLNNC**  Output of all non-neighbor connections:

1. The local grid non-neighbor connections, and connections between the global and local grids, are written to the Print file.

2. As 1, but the connections between the global and local grids are also written to the Debug file.

3. As 2, but the connections written to the Debug file have the local grids stacked in the z-direction so that they do not overlay one another.

   The output to the Debug file is useful when converting an ECLIPSE 100 LGR to a traditional in place refinement.

57  **EXTHOST**  Output host cells for unstructured (PEBI) LGRs.

---

**Note:** The GRID section data will only be output for local refined grids if it is possible to output similar data for global cells (refer to the E300 availability column in the RPTGRID keyword description). It is currently not possible to output GRID section information using this keyword for a parallel run.

---

**Note:** COALV and PORVF are not available in a PARALLEL run.

The data entered with keyword RPTGRIDL applies to all local grids. The keyword needs only to be specified once. If specified more than once, the last specification applies.

If RPTGRIDL is not used, there will be no GRID section data output for local grids.

Output requested using the keyword RPTGRID applies only to the global grid.

In ECLIPSE 100, grid-valued output values in certain types of cell have the decimal point replaced with a special character for ease of identification. See 3.31 for details.
## Controls on output to the HMD file

The keyword should be followed some or all of the following integer items, which control the amount of data written to the HMD file (see "Gradient Option" in the ECLIPSE Technical Description). Values greater than zero generally reduce the amount of output. The data should be terminated with a slash (/). Repeat counts (for example, 3*0) can be used if required but with no spaces before or after the asterisk.

<table>
<thead>
<tr>
<th>Item</th>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Item 1</td>
<td>If set = 0</td>
<td>Derivatives for all parameters are output.</td>
</tr>
<tr>
<td></td>
<td>If set &gt; 0</td>
<td>No derivatives are output and all derivative calculations are switched off (equivalent to NOHMD).</td>
</tr>
<tr>
<td></td>
<td>DEFAULT: 0</td>
<td></td>
</tr>
<tr>
<td>Item 2</td>
<td>If set = 0</td>
<td>Derivatives are output at each simulator timestep</td>
</tr>
<tr>
<td></td>
<td>If set &gt; 0</td>
<td>Derivatives will be output only at report times (RPTONLY)</td>
</tr>
<tr>
<td></td>
<td>DEFAULT: 0</td>
<td></td>
</tr>
<tr>
<td>Item 3</td>
<td>If set = 0</td>
<td>Derivatives will be output for all producers, injectors and any shut or stopped wells.</td>
</tr>
<tr>
<td></td>
<td>If set &gt; 0</td>
<td>Derivatives will only be output for active producers (a producer is active if it is open or stopped, but not shut.)</td>
</tr>
<tr>
<td></td>
<td>DEFAULT: 0</td>
<td></td>
</tr>
<tr>
<td>Item 4</td>
<td>If set = 0</td>
<td>The values of well rates, pressures and so on are written in addition to the derivatives</td>
</tr>
<tr>
<td></td>
<td>If set &gt; 0</td>
<td>The values of well rates, pressures and so on ('VALUES' record in the .HMD file) are not output.</td>
</tr>
<tr>
<td></td>
<td>DEFAULT: 0</td>
<td></td>
</tr>
<tr>
<td>Item 5</td>
<td>If set = 0</td>
<td>The 'VALUES' and 'DERIVS' records in the .HMD file will be double precision.</td>
</tr>
<tr>
<td></td>
<td>If set &gt; 0</td>
<td>The 'VALUES' and 'DERIVS' records in the .HMD file will be single precision.</td>
</tr>
<tr>
<td></td>
<td>DEFAULT: 0</td>
<td></td>
</tr>
<tr>
<td>Item 6</td>
<td>If set = 0</td>
<td>Group, field and cumulative production gradients may not be requested</td>
</tr>
<tr>
<td></td>
<td>If set &gt; 0</td>
<td>Cumulative gradients (WOPT, WWPT and WGPT for example) may be requested using the GRADWELL keyword. Group and field gradients (GOPR, GWPR and GGPR for example) may be requested using GRADGRUP keyword.</td>
</tr>
<tr>
<td></td>
<td>DEFAULT: 0</td>
<td></td>
</tr>
</tbody>
</table>

*Table 3.32: RPTHMD item settings*

At a restart when SKIPREST has been read, RPTHMD is then processed while skipping any data up to the restart time, analogously to the RPTSCHED and RPTRST keywords.

### Example

Only output derivatives for production wells at report times

```plaintext
RPTHMD 0 1 1 1 /
```
Configure output of derivatives for groups

The keyword can be used to switch on or off the output of gradient information to the HMD file for individual groups (see "Gradient Option" in the ECLIPSE Technical Description). This is normally used by the SimOpt program.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Group name.
   A well name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record. Field quantities are obtained by specifying ‘FIELD’ as the group name.

2. Flag to switch the output for the specified group (or groups) on or off in the gradient HMD file.
   - **ON** Switch on the output of derivatives for this group if the output has previously been switched off.
   - **OFF** Switch off the output of derivatives for this group if the output was previously switched on.

   **DEFAULT**: ON

In order to use any group quantities, the 6th switch of the RPTHMD keyword should be set positive.

When item 2 is set to ON, the group gradient quantities written to the HMD file are configured by the GRADGRUP keyword.

If the keyword is entered without any data records (that is followed only by blank record and terminated by a slash) then every group will be output until the flag is reset.

**Examples**

**Example 1**

```
RPTHMG
'TARBERT' OFF /
'NESS' ON /
'FIELD' ON /
/
```

**Example 2**

Switch on output for every group

```
RPTHMG /
```
Configure output of derivatives for wells

The keyword can be used to switch on or off the output of gradient information to the HMD file for individual wells (see "Gradient Option" in the ECLIPSE Technical Description). This is normally used by the SimOpt program.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Flag to switch the output for the specified well (or wells) on or off in the gradient HMD file.
   ON Switch on the output of derivatives for this well if the output has previously been switched off.
   OFF Switch off the output of derivatives for this well if the output was previously switched on.

   DEFAULT: ON

3. For ECLIPSE 100, this flag switches the RFT gradient output for the specified well (or wells) on or off in the RFT file.
   ON Switch on the output of RFT derivatives for this well if the output has previously been switched off.
   OFF Switch off the output of RFT derivatives for this well if the output was previously switched on.

   DEFAULT: OFF

The third item requesting RFT gradients is not available in ECLIPSE 300.

Note that any actions specified in the RPTHMD keyword take priority over those specified using RPTHMW. Thus, for example, if output for injection wells is switched off by item 3 in RPTHMD, then any action in RPTHMW will be ignored for injection wells.

When item 2 is set to ON, the well gradient quantities written to the HMD file are configured by the GRADWELL keyword.

When item 3 is set to ON, the connection gradient quantities written to the RFT file are configured by the GRADRFT keyword.

If the keyword is entered without any data records (that is followed only by blank record and terminated by a slash) then every well will be output until the flag is reset.

Examples

Example 1

| RPTHMW | PRODUCER | OFF    | OFF / |

Keywords
Example 2

Switch on output for every well

```
'RPTHMW
```
Controls on output from GRID and EDIT sections to INIT file

The keyword should be followed by a string of mnemonics to request output of GRID and EDIT data to the .INIT (or .FINIT) file. The appropriate keywords used to input some GRID and EDIT data, such as DX and DY, may be used as mnemonics. Aliases may be used; for example DR may be used in place of DX and DTHETA may be used in place of DY.

By default, some arrays will always be output depending on the model definition. For instance, to name a few examples, the minimum cell rock volume (MINRVV) will always be output for a THERMAL run; FLUXNUM, MULTNUM and OPERNUM will always be output if this information is present in the dataset.

The data should be terminated by a slash (/).

When the keyword is specified along with a series of mnemonics, only PORV and the mnemonics specified will be output to the .INIT or .FINIT file. This has been extended to include Cartesian LGRs, radial LGRs and nested LGRs.

The following table lists the mnemonics available. The mnemonics available for each simulator are identified by an x in the appropriate column.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>AREAX</td>
<td>Output of X direction Area</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>(Velocity dependent flow option;</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Velocity dependent relative permeabilities (E300))</td>
<td></td>
<td></td>
</tr>
<tr>
<td>AREAY</td>
<td>Output of Y direction Area</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>(Velocity dependent flow option;</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Velocity dependent relative permeabilities (E300))</td>
<td></td>
<td></td>
</tr>
<tr>
<td>AREAZ</td>
<td>Output of Z direction Area</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>(Velocity dependent flow option;</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Velocity dependent relative permeabilities (E300))</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BTOBALFV</td>
<td>Output of Dual Porosity block-to-block matrix-fracture coupling</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>multiplier</td>
<td></td>
<td></td>
</tr>
<tr>
<td>COALV</td>
<td>Output of grid block coal volumes (Coal Bed Methane)</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DEPTH</td>
<td>Output of grid block center depths</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DIFFX</td>
<td>Output of X direction diffusivities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DIFFY</td>
<td>Output of Y direction diffusivities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DIFFZ</td>
<td>Output of Z direction diffusivities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DIFFMX</td>
<td>Output of X direction diffusivity multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DIFFMY</td>
<td>Output of Y direction diffusivity multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DIFFMZ</td>
<td>Output of Z direction diffusivity multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DIFFMMF</td>
<td>Output of Matrix/Fracture diffusivity multipliers</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DIFFMX-</td>
<td>Output of X- direction diffusivity multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>-----------</td>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>DIFFMY-</td>
<td>Output of Y- direction diffusivity multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DIFFMZ-</td>
<td>Output of Z- direction diffusivity multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DR</td>
<td>Output of R direction grid block sizes</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DTHETA</td>
<td>Output of THETA direction grid block sizes</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DX</td>
<td>Output of X direction grid block sizes</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DY</td>
<td>Output of Y direction grid block sizes</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DZ</td>
<td>Output of Z direction grid block sizes</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>DZMTRXV</td>
<td>Output of D_{zmatr} values</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>DZMTRXM</td>
<td>Output of net DZ values</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>FRACTURE</td>
<td>Output of fracture propagation data (fracture modeling only)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>HEATCR</td>
<td>Output of rock heat capacity</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>HEATCRT</td>
<td>Output of temperature dependence of the rock heat capacity</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>HEATTX</td>
<td>Output of X direction Thermal transmissibilities</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>HEATTY</td>
<td>Output of X direction Thermal transmissibilities</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>HEATTZ</td>
<td>Output of X direction Thermal transmissibilities</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>IONROCK</td>
<td>Rock ion exchange capacity</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>LX</td>
<td>Output of matrix block size in X direction</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>LY</td>
<td>Output of matrix block size in Y direction</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>LZ</td>
<td>Output of matrix block size in Z direction</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MIDS</td>
<td>Output of midpoint depths</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MINPVV</td>
<td>Output of cells with pore volume less than the input minimum values</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MULTPV</td>
<td>Output of the pore volume multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MULTX</td>
<td>Output of X direction transmissibility multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MULTY</td>
<td>Output of Y direction transmissibility multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MULTZ</td>
<td>Output of Z direction transmissibility multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MULTX-</td>
<td>Output of X- direction transmissibility multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MULTY-</td>
<td>Output of Y- direction transmissibility multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MULTZ-</td>
<td>Output of Z- direction transmissibility multipliers</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>NTG</td>
<td>Output of grid block net-to-gross ratios</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PERMX</td>
<td>Output of X direction permeabilities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PERMY</td>
<td>Output of Y direction permeabilities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>--------------</td>
<td>-------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>PERMZ</td>
<td>Output of Z direction permeabilities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PORO</td>
<td>Output of grid block porosities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PORV</td>
<td>Output of grid block pore volumes</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PORVF</td>
<td>Output of grid block fluid pore volumes (Coal Bed Methane option)</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>ROCKFRAC</td>
<td>Output of grid block rock volume fractions (Coal Bed Methane option)</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>ROCKDEN</td>
<td>Output of rock density</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ROCKV</td>
<td>Output of rock volumes</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SIGMAGDV</td>
<td>Output of SIGMAGD values (gravity drainage option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SIGMAV</td>
<td>Output of Sigma values</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SIGMA*K</td>
<td>Output of the processed $f_x \cdot K_x \cdot l_x^2 + f_y \cdot K_y \cdot l_y^2 + f_z \cdot K_z \cdot l_z^2$ term for the matrix-fracture transmissibility in dual porosity runs, if item 5 of the LTOSIGMA keyword is set to ALL.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>THCGAS</td>
<td>Output of Thermal conductivity for gas phase</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>THCOIL</td>
<td>Output of Thermal conductivity for oil phase</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>THCONR</td>
<td>Output of thermal rock and fluid conductivities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>(For ECLIPSE 300 only, this header will be renamed THCAVE, similar to RPTGRID keyword)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>THCONSF</td>
<td>Output saturation dependence of conductivity</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>THCSOLID</td>
<td>Output of Thermal conductivity for solid phase</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>THCWATER</td>
<td>Output of Thermal conductivity for water phase</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>TOPS</td>
<td>Output of grid block top depths</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>TRANX</td>
<td>Output of X direction transmissibilities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>TRANY</td>
<td>Output of Y direction transmissibilities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>TRANZ</td>
<td>Output of Z direction transmissibilities</td>
<td>x</td>
<td>x</td>
</tr>
</tbody>
</table>

*Table 3.33: RPTINIT output controls*

**Example**

```
RPTINIT
DZ TRANX PORO PERMY /
```
Generates a grid array of isolated reservoir numbers

This keyword requests ECLIPSE to generate a grid array of isolated reservoir region numbers, which it outputs to the Debug file. The array can then be extracted and included in the input data file under the keyword ISOLNUM, to activate the efficient solution technique for independent reservoir regions (see “Independent reservoir regions” in the ECLIPSE Technical Description).

ECLIPSE determines the extent of each region of the reservoir (if any) that is isolated from the rest of the grid by examining the flow connections between grid blocks (normal transmissibility, non-neighbor connections, and multiply connected wells). If the model contains more than one independent reservoir region, it may be solved more efficiently by activating the Independent Reservoir Regions facility with the keyword ISOLNUM.

The RPTISOL keyword has no associated data.

Example

```
RPTISOL
```
**RPTONLY**

**Limits the frequency of summary output**

This keyword requests that summary data be written to the Summary file only at report times. The default, if the RPTONLY keyword is not present (and it was not requested in a restart file), is to write the summary information at every timestep.

As RPTONLY is available in the SCHEDULE section, the setting can change during the simulation. The setting from a base run will be preserved in a restart run. To change the setting in a restart run, the RPTONLY (or RPTONLYO) keyword must be specified at or after the restart time. Any RPTONLY (or RPTONLYO) keywords in the SUMMARY section, or in the SCHEDULE section before the restart time, will be ignored in a restart run.

This facility can be useful when the size of the Summary files is important, for example when ECLIPSE is run at a remote site and the Summary files are transferred via a slow line. This restriction on the output can be turned off by using the RPTONLYO keyword.

The RPTONLY keyword has no associated data.

**Example**

```plaintext
RPTONLY
```

---

**Keywords**

1894
RPTONLY  Turns off RPTONLY restricted summary output

This keyword requests that summary data be written to the Summary file at every timestep. This is the default situation unless overridden in a restart file. This keyword need, therefore, only be used when RPTONLY has previously been used and it is wished to turn off this behavior later in the run.

As RPTONLY is available in the SCHEDULE section, the setting can change during the simulation. The setting from a base run will be preserved in a restart run. To change the setting in a restart run, the RPTONLY (or RPTONLY) keyword must be specified at or after the restart time. Any RPTONLY (or RPTONLY) keywords in the SUMMARY section, or in the SCHEDULE section before the restart time, will be ignored in a restart run.

This facility can be useful when the size of the Summary files is important but more detail is needed at certain times.

The RPTONLY keyword has no associated data.

Example

```
RPTONLY
```
Optimization reporting options

This keyword is used to request the detail of reporting from the reservoir optimization option. The presence of this keyword is entirely optional.

The keyword is followed by a list of mnemonics terminated by a slash (/).

Available mnemonics are summarized in the table.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>SEPARATE</td>
<td>Output separate summary and restart files for each optimization iteration. These have the form BASE_01.xxx, BASE_02.xxx, and so on where BASE.DATA is the base dataset, and _01,_02, ... label the optimization iteration.</td>
</tr>
<tr>
<td>CLRFILES</td>
<td>Removes SEPARATE results files from any previous optimization for this model if necessary (that is, if the current optimization is about to overwrite some or all of them). The current optimization will be considered about to overwrite previous optimization results files if it produces separate results files (so the SEPARATE mnemonic must be currently selected) with a formatted and unified status that matches those of a previous optimization. This removal is useful in the situation where the current optimization takes fewer iterations than the previous optimization, but the presence of the old additional iteration results files leads to misleading interpretations of the results (for example, when plotting all the iterations results in ECLIPSE Office).</td>
</tr>
</tbody>
</table>

Example

```
RPTOPT
-- Output separate summary and restart files for each
-- optimization iteration, and clear previous results files
`SEPARATE` `CLRFILES` /
```
RPTPRINT Control printed simulation reports

This keyword controls the output of printed reports. It has the following integer data items, each of which controls a printed report. It may be specified any number of times in the SCHEDULE section.

   DEFAULT: 1

2. Report fluid in place for field.
   If this is set to 2 or greater, when used in combination with item 3 or 11 or 12, reports the values for the additional set of fluid-in-place regions apart from the standard set.
   DEFAULT: 1

   If this is set to 2 or greater, the fluid in place by region will include a separate flash of reservoir oil and reservoir gas to stock tank conditions.
   To report values for additional set of fluid-in-place regions, item 2 of this keyword need to be set to 2 and the value of this item to be set to 1 or 2 as desired.
   DEFAULT: 1

4. Report status of groups.
   DEFAULT: 1

5. Not used at present

   DEFAULT: 1

   DEFAULT: 1

8. Report solution data, as specified by mnemonics in RPTSCHED.
   DEFAULT: 1

   DEFAULT: 0

    DEFAULT: 0

    To report values for additional set of fluid-in-place regions, item 2 of this keyword need to be set to 2 and value of this item to be set to 1.
    DEFAULT: 0

    To report values for additional set of fluid-in-place regions, item 2 of this keyword need to be set to 2 and value of this item to be set to 1.
   DEFAULT: 0
   DEFAULT: 0
15. Reserved for use by FrontSim.
16. Reserved for use by FrontSim.
17. Reserved for use by FrontSim.
18. Report status of network
   DEFAULT: 1

Each item may be set to 0 (or a negative number), to turn off the appropriate type of report, or to 1 (or a positive number), to turn it on. **RPTSCHED** in ECLIPSE 300 is a method of specifying which solution arrays (PRESSURE, SWAT, SGAS, and VOIL for example) are to be printed out under the control of the item in RPTPRINT.

**Example**

Request step summary, fluid in place reports, region fluid in place reports, group and well reports, and output of the solution arrays specified by mnemonics with **RPTSCHED**. Well completion reports, convergence reports and well potentials turned off.

```
RPTPRINT 1 1 1 1 0 1 0 1 0 0 /
```
RPTPROPS  Controls on output from PROPS section

The keyword should be followed by a list of mnemonics which control the output of PROPS section data to the Print file.

The list should be terminated by a slash (/).

In ECLIPSE 100

- Entering the mnemonic NOTHING clears all the reporting flags.
- For additional control over the output, integer values greater than one may be assigned to selected mnemonics using the syntax:

  mnemonic=integer

  where there must be no spaces either side of the equals sign.

Older ECLIPSE 100 data sets may not use mnemonics but integer controls. The meaning of these is documented in "RPTPROPS" in the ECLIPSE Technical Description.

The following table lists the mnemonics available. Mnemonics available in each simulator are identified by an x in the respective column.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALSURFST</td>
<td>Outputs table of water-oil surface tension multipliers as a function of alkaline concentration</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>AQUATAB</td>
<td>Outputs the Carter-Tracy influence tables</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DENSITY, GRAVITY, SDENSITY, BDENSITY</td>
<td>Output of surface densities/gravities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DIFFC or DIFFCOAL</td>
<td>Output of the diffusion data</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DIFFCBM</td>
<td>Output of diffusion data (Coal Bed Methane)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DETAILVD</td>
<td>Output of the DETAILVD tables</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ENDPT</td>
<td>Output of endpoint scaling values</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ENDPTVD</td>
<td>Output of endpoint versus depth tables</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>EPSODD3P</td>
<td>Output of end-point scaling control table for the ODD3P option</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ESPNODE</td>
<td>Output of effective salinity nodes for Polymer Flood Model</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ESSNODE</td>
<td>Output of effective salinity nodes for Surfactant Model</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>HHERCHBL</td>
<td>Output of Herschel-Bulkley data (fracture modeling only)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FOAM</td>
<td>Output of the foam properties</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FRACMOVE</td>
<td>Output of fracture propagation (fracture modeling only)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>GINODE</td>
<td>Output of Gi-model saturated fluid properties</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>HSGRO</td>
<td>Output of hysteresis gas residual end-point saturation for gas-oil system for the ODD3P option</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>--------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>HSGRW</td>
<td>Output of hysteresis gas residual end-point saturation for gas-water system for the <strong>ODD3P</strong> option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>HSORG</td>
<td>Output of hysteresis oil residual end-point saturation for oil-gas system for the <strong>ODD3P</strong> option</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>HSORW</td>
<td>Output of hysteresis oil residual end-point saturation for oil-water system for the <strong>ODD3P</strong> option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>HSWRG</td>
<td>Output of hysteresis water residual end-point saturation for water-gas system for the <strong>ODD3P</strong> option</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>HSWRO</td>
<td>Output of hysteresis water residual end-point saturation for water-oil system for the <strong>ODD3P</strong> option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>IKRWH</td>
<td>Output of water relative permeabilities at maximum water saturation for WAG hysteresis three-phase water relative permeabilities when end-point scaling has been specified.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>IKRWRH</td>
<td>Output of water relative permeabilities at residual oil for WAG hysteresis three-phase water relative permeabilities when end-point scaling has been specified.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>KRG</td>
<td>Output of maximum gas relative permeabilities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>KRGR</td>
<td>Output of gas relative permeabilities at residual</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>KRO</td>
<td>Output of maximum oil relative permeability</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KGOR</td>
<td>Output of oil relative permeabilities at residual</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>KRW</td>
<td>Output of maximum water relative permeabilities</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>KRWH</td>
<td>Output of water relative permeabilities at maximum water saturation for WAG hysteresis two-phase water relative permeabilities when end-point scaling has been specified.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>KRWR</td>
<td>Output of water relative permeabilities at residual</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>KRWRH</td>
<td>Output of water relative permeabilities at residual oil for WAG hysteresis two-phase water relative permeabilities when end-point scaling has been specified.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>LANGMEXT</td>
<td>Output of the extended Langmuir Isotherms data</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>LANGMPL</td>
<td>Output of the pressure scaling factor for the adsorption capacity for each grid cell.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>LANGMULT</td>
<td>Output of the Langmuir multipliers</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>LANGMUIR, LANGSOLV, COALADS</td>
<td>Output of the Langmuir Isotherms</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>LEAKOFF</td>
<td>Output of fracture fluid loss (fracture modeling only)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MLANG</td>
<td>Output of the maximum surface gas concentration</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MLANGSLV</td>
<td>Output of the maximum surface solvent concentration</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MISC, PMISC</td>
<td>Output of the miscibility function and the pressure miscibility function</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>OVERBURD</td>
<td>Overburden pressure data</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>--------------</td>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>PCG</td>
<td>Output of scaled maximum gas capillary pressures (provided data is supplied via the PCG keyword), or the scaling factor for the JFUNC keyword</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PCODD3P</td>
<td>Output of capillary pressure calculation selection tables for the ODD3P option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PCODD3PG</td>
<td>Output of capillary pressure calculation gas saturation function for the ODD3P option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PCODD3PW</td>
<td>Output of capillary pressure calculation water saturation function for the ODD3P option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PCW</td>
<td>Output of scaled maximum water capillary pressures (provided data is supplied via the PCW keyword), or the scaling factor for the JFUNC keyword</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PLYADS, PLYMAX, PLYROCK, PLYSHEAR, PLYDHFIL, PLYVISC, PLYVISCS, PLYVISCT, PLYVISCST, SALTNODE, TEMPNODE, PLMXPAR</td>
<td>Output of Polymer Flood Model properties. If the control is set to 2 for a mnemonic starting PL, the output includes values for the grid-valued keywords PLYKRF, PLYRMDEN and PLYCAMAX.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PLYVISC</td>
<td>Output of water phase viscosity multipliers supplied by the PLYVISC keyword</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PLYVISCS</td>
<td>Output of water phase viscosity multipliers supplied by the PLYVISCS keyword</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PSGRO</td>
<td>Output of primary gas residual end-point saturation for gas-oil system for the ODD3P option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PSGRW</td>
<td>Output of primary gas residual end-point saturation for gas-water system for the ODD3P option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PSORG</td>
<td>Output of primary oil residual end-point saturation for oil-gas system for the ODD3P option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PSORW</td>
<td>Output of primary oil residual end-point saturation for oil-water system for the ODD3P option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PSWRG</td>
<td>Output of primary water residual end-point saturation for water-gas system for the ODD3P option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PSWRO</td>
<td>Output of primary water residual end-point saturation for water-oil system for the ODD3P option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PVDG, PVTG</td>
<td>Output of gas PVT tables</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PVDO, PDTO</td>
<td>Output of oil PVT tables</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PVDS</td>
<td>Output of Solvent PVT properties</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>--------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>PVTW</td>
<td>Output of water PVT tables</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PVTWSALT</td>
<td>Output of brine properties</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>ROCK</td>
<td>Output of rock properties</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ROCKTAB</td>
<td>Output of rock properties</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ROCKTABH</td>
<td>Output of rock properties (Hysteretic option only)</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SGCRI</td>
<td>Output of scaled critical gas saturations</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SGPN</td>
<td>Output of gas saturation tables</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SGP3</td>
<td>Output of gas saturation tables for the IKU3P and ODD3P options. Output of three-phase gas relative permeability table for the IKU3P option.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SGL</td>
<td>Output of scaled connate gas saturations</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SGU</td>
<td>Output of scaled maximum gas saturations</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SOF2, SOF3, TLMIXPAR, SOURNMIS</td>
<td>Output of oil saturation tables, three-phase relative permeability tables, Todd-Longstaff mixing parameter, Miscible residual oil saturation tables. If control is set = 2, the output includes a print of the residual oil saturation as a function of water and gas saturations.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SOGCR</td>
<td>Output of scaled critical oil-in-gas saturations</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SOWCR</td>
<td>Output of scaled critical oil-in-water saturations</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SOLU</td>
<td>Output of aqueous phase property data (CO2SOL and GASSOL options)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SPECHEAT, VISCVT</td>
<td>Output of the temperature-dependent properties.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SSFN</td>
<td>Output of Solvent/Gas relative permeabilities.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>STOG, STOW</td>
<td>Output of Surface Tension properties.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>STONE1EX</td>
<td>Output of Stone 1 exponent values.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SURF</td>
<td>Output of surfactant properties including: surfactant solution viscosities (SURFVISC), adsorption (SURFADS), water-oil surface tension (SURFST or SURFSTES), surfactant capillary de-saturation functions (SURFCAPD), adsorbed surfactant concentration vs fractional saturation functions (SURFADDW), surfactant rock properties (SURFROCK) and salinity nodes (ESSNODE).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SURFSTES</td>
<td>Outputs table of water-oil surface tension as function of surfactant concentration and effective salinity</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SWCR</td>
<td>Output of scaled critical water saturations.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SWFN</td>
<td>Output of water saturation tables.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SWF3</td>
<td>Output of water saturation tables for the IKU3P and ODD3P options. Output of three-phase water relative permeability table for the IKU3P option.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SWL</td>
<td>Output of scaled connate water saturations.</td>
<td>x</td>
<td>x</td>
</tr>
</tbody>
</table>
### Mnemonic Output E100 E300

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>SWU</td>
<td>Output of scaled maximum water saturation.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>TRACER</td>
<td>Output of passive fluid tracer names.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>TRADS, TRDCY, TRROCK</td>
<td>Output of environmental tracer properties.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>VDKRG</td>
<td>Velocity dependent relative permeability data for gas.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>VDKRO</td>
<td>Velocity dependent relative permeability data for oil.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>VEFRACV, VEFRAC</td>
<td>Output of Vefrac values.</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

**Table 3.35: RPTPROPS output controls**

**ECLIPSE 100**

If a value equal to 5 is used for any (or all) of SOF3, SWFN or SGFN, the saturation tables are printed at saturation interval of 5 percent. These values are printed in addition to the user-specified values and are calculated using the user-specified values. Similarly, if a value equal to 5 is used for the fourth, the fifth and the sixth control, the PVT tables are printed at a pressure interval of 200 psi (FIELD), 10 bars (METRIC) or 10 atm (LAB). As before, these values are printed in addition to the user-specified values.

**ECLIPSE 100**

If the PVTO table output option is specified, the interpolated versions of the tables will also be printed if API tracking has been selected.

**ECLIPSE 100**

For models, for which end-point scaling has been specified, if the LOWSALT option has been specified, the low salinity (oil-wet) end-points, for example, LSWL, can selected for reporting. Similarly, if the SURFACTW option has been specified, the (high salinity) water-wet end-points, for example, HWSWL, can selected for reporting. If both of these options have been specified, the low salinity water-wet end-points, for example, LWSWL may also be selected for reporting.

**ECLIPSE 300**

If directional end point scaling is active via the DIRECT and IRREVERS items of the ENDScale keyword, then each of the preceding set of mnemonics (except for the capillary pressure mnemonics PCG and PCW) can be followed by either X, Y or Z (for the DIRECT option) and X-, Y- or Z- (if the IRREVERS option is active), for example KRGRZ-.

**ECLIPSE 100**

Grid-valued output values in certain types of cell have the decimal point replaced with a special character for ease of identification. See Table 3.31 for details.

### Examples

#### Example 1 (ECLIPSE 300)

```
RPTPROPS
SWFN SGFN PVTO PVTW PVDG /
```

#### Example 2 (ECLIPSE 100)

Changing the level of output

```
RPTPROPS
  SOF3=5 SWFN=5 SGFN=5 /
```
Controls on output from REGIONS section

The keyword should be followed by a list of mnemonics which control the output of REGIONS section data to the Print file.

The list should be terminated by a slash (/).

In ECLIPSE 100

- Entering the mnemonic NOTHING clears all the reporting flags.
- For additional control over the output, integer values greater than one may be assigned to selected mnemonics using the syntax:
  \[ \text{mnemonic}=\text{integer} \]
  where there must be no spaces either side of the equals sign.
- Optionally, for back compatibility, a list of integers (terminated with a slash) may be used to indicate the required output. A value less than or equal to zero switches off the corresponding output. Values greater than zero switch the output on. Repeat counts (for example 3*0) can be used if required (with no spaces before or after the asterisk). If fewer than 26 integers are read, the remainder are left unchanged. The integer list cannot be mixed with mnemonics.

The following table lists the mnemonics available, and also their equivalent integers for the ECLIPSE 100 integer lists. Mnemonics without an integer in the E100 column are not available in ECLIPSE 100. Mnemonics available in ECLIPSE 300 are identified by an x in the E300 column.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100 Integer Control</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALPHANUM</td>
<td>Output of ALPHA region numbers</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DSTNUM</td>
<td>Output of tertiary hysteresis saturation region numbers for the ODD3P option</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ENDNUM</td>
<td>Output of endpoint v depth region numbers</td>
<td>19</td>
<td>x</td>
</tr>
<tr>
<td>EOSNUM</td>
<td>Output of equation of state region numbers</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>EQLNUM</td>
<td>Output of equilibration region numbers</td>
<td>3</td>
<td>x</td>
</tr>
<tr>
<td>FIPNUM</td>
<td>Output of fluid in place region numbers</td>
<td>4</td>
<td>x</td>
</tr>
<tr>
<td>FIPOWG</td>
<td>Output original oil, water and gas zones</td>
<td>28</td>
<td></td>
</tr>
<tr>
<td>FRACNUM</td>
<td>Output of fracture region numbers (fracture modeling only)</td>
<td>31</td>
<td></td>
</tr>
<tr>
<td>HMxxxxxx</td>
<td>Output of gradient parameter regions when the Gradient Option is in use. (If set &gt;1, the parameter regions will be written to the Initial file when the INIT keyword is specified, this only applies to ECLIPSE 100).</td>
<td>25</td>
<td>x</td>
</tr>
<tr>
<td>HWSNUM</td>
<td>Output of (high salinity) water-wet saturation region numbers when the SURFACTW option has been specified. This keyword is an alias for the SURFNUM keyword.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100 Integer Control</td>
<td>E300</td>
</tr>
<tr>
<td>------------</td>
<td>------------------------------------------------------------------------</td>
<td>----------------------</td>
<td>------</td>
</tr>
<tr>
<td>IMBNUM</td>
<td>Output of imbibition region numbers</td>
<td>11</td>
<td>x</td>
</tr>
<tr>
<td>IMBNUMMF</td>
<td>Output of matrix/fracture imbibition regions</td>
<td>23</td>
<td></td>
</tr>
<tr>
<td>IMBNUMX</td>
<td>Output of X direction imbibition region numbers</td>
<td>12</td>
<td>x</td>
</tr>
<tr>
<td>IMBNUMX−</td>
<td>Output of -X direction imbibition region numbers</td>
<td>13</td>
<td>x</td>
</tr>
<tr>
<td>IMBNUMY</td>
<td>Output of Y direction imbibition region numbers</td>
<td>14</td>
<td>x</td>
</tr>
<tr>
<td>IMBNUMY−</td>
<td>Output of -Y direction imbibition region numbers</td>
<td>15</td>
<td>x</td>
</tr>
<tr>
<td>IMBNUMZ</td>
<td>Output of Z direction imbibition region numbers</td>
<td>16</td>
<td></td>
</tr>
<tr>
<td>IMBNUMZ−</td>
<td>Output of -Z direction imbibition region numbers</td>
<td>17</td>
<td></td>
</tr>
<tr>
<td>ISTNUM</td>
<td>Output of secondary hysteresis saturation region numbers for the <strong>ODD3P</strong> option</td>
<td></td>
<td></td>
</tr>
<tr>
<td>LSNUM</td>
<td>Output of low salinity (oil-wet) saturation region numbers when the <strong>LOWSALT</strong> option has been specified. This keyword is an alias for the <strong>LWSLTNUM</strong> keyword.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>LWSLTNUM</td>
<td>Output of low salinity (oil-wet) saturation region numbers when the <strong>LOWSALT</strong> option has been specified. This keyword is an alias for the <strong>LSNUM</strong> keyword.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>LWSNUM</td>
<td>Output of low salinity water-wet saturation region numbers when the <strong>SURFACTW</strong> and the <strong>LOWSALT</strong> options have been specified. This keyword is an alias for the <strong>LSLTWNUM</strong> keyword.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>LSLTWNUM</td>
<td>Output of low salinity water-wet saturation region numbers when the <strong>SURFACTW</strong> and the <strong>LOWSALT</strong> options have been specified. This keyword is an alias for the <strong>LWSNUM</strong> keyword.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRNUMMF</td>
<td>Output of matrix/fracture saturation regions</td>
<td>22</td>
<td></td>
</tr>
<tr>
<td>KRNUMX</td>
<td>Output of X direction ( K_r ) region numbers</td>
<td>5</td>
<td>x</td>
</tr>
<tr>
<td>KRNUMX−</td>
<td>Output of -X direction ( K_r ) region numbers</td>
<td>6</td>
<td>x</td>
</tr>
<tr>
<td>KRNUMY</td>
<td>Output of Y direction ( K_r ) region numbers</td>
<td>7</td>
<td>x</td>
</tr>
<tr>
<td>KRNUMY−</td>
<td>Output of -Y direction ( K_r ) region numbers</td>
<td>8</td>
<td>x</td>
</tr>
<tr>
<td>KRNUMZ</td>
<td>Output of Z direction ( K_r ) region numbers</td>
<td>9</td>
<td>x</td>
</tr>
<tr>
<td>KRNUMZ−</td>
<td>Output of -Z direction ( K_r ) region numbers</td>
<td>10</td>
<td>x</td>
</tr>
<tr>
<td>MISCNUM</td>
<td>Output of miscibility region numbers</td>
<td>21</td>
<td>x</td>
</tr>
<tr>
<td>PMANUM</td>
<td>Pressure maintenance region numbers</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSTNUM</td>
<td>Output of primary saturation region numbers for the <strong>ODD3P</strong> option</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PVTNUM</td>
<td>Output of PVT region numbers</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>RESIDNUM</td>
<td>Output of VE residual flow region numbers</td>
<td>26</td>
<td></td>
</tr>
<tr>
<td>ROCKNUM</td>
<td>Output of Rock Compaction region numbers</td>
<td>18</td>
<td>x</td>
</tr>
<tr>
<td>SATNUM</td>
<td>Output of saturation region numbers</td>
<td>2</td>
<td>x</td>
</tr>
</tbody>
</table>

**Keywords**

1905
<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100 Integer Control</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>SDRGO</td>
<td>Output of gas phase saturation direction indicators for gas-oil system for the ODD3P option</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SDRGW</td>
<td>Output of gas phase saturation direction indicators for gas-water system for the ODD3P option</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SDROG</td>
<td>Output of oil phase saturation direction indicators for oil-gas system for the ODD3P option</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SDROW</td>
<td>Output of oil phase saturation direction indicators for oil-water system for the ODD3P option</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SDRWG</td>
<td>Output of water phase saturation direction indicators for water-gas system for the ODD3P option</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SDRWO</td>
<td>Output of water phase saturation direction indicators for water-oil system for the ODD3P option</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SURFNUM</td>
<td>Output of the surfactant saturation regions</td>
<td></td>
<td>24</td>
</tr>
<tr>
<td>SURFWNUM</td>
<td>Output of (high salinity) water-wet saturation region numbers when the SURFACTW option has been specified. This keyword is an alias for the HWSNUM keyword.</td>
<td></td>
<td>27</td>
</tr>
<tr>
<td>WH2NUM</td>
<td>Output of two-phase water relative permeability saturation region numbers for the WAGHYSTR option.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>WH3NUM</td>
<td>Output of three-phase water relative permeability saturation region numbers for the WAGHYSTR option.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TNUM</td>
<td>Output of tracer versus depth region numbers</td>
<td></td>
<td>20</td>
</tr>
<tr>
<td>TRACKREG</td>
<td>Output of enhanced TRACK region numbers</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

Table 3.36: RPTREGS output controls

**Note:** For ECLIPSE 100, LWSLTNUM and its alias LSNUM, SURFWNUM and its alias HWSNUM, and LSLOTNUM and its alias LWSNUM are interchangeable, with the latter having been introduced to provide a closer correspondence between the saturation region names and the associated end-point scaling array names. The title provided in the PRT file will correspond with whichever name is entered; if both names are entered, the title will be determined by the later of the two names in the list.

### Examples

#### Example 1

```plaintext
RPTREGS
FIPNUM ENDMEM SATNUM /
```

#### Example 2 (ECLIPSE 100)

The old integer format:

```plaintext
RPT5*0 1 /
```
**RPTRST**

**Controls on output to the RESTART file**

The keyword should be followed by a list of mnemonics which control the output of data to the Restart file. The list should be terminated by a slash (/). Note that no restart files will be written if no RPTRST input keywords or mnemonics are specified.

For additional control over the output, integer values greater than one may be assigned to selected mnemonics using the syntax:

```
mnemonic=integer
```

where there must be no spaces either side of the equals sign.

*Note:* Older ECLIPSE 100 data sets may not use mnemonics but integer controls. The meaning of these is documented in "RPTRST" in the ECLIPSE Technical Description.

*Note:* Cannot be used in the SOLUTION section for ECLIPSE 100 parallel fast restart runs. It must instead be placed at the start of the SCHEDULE section.

---

**ECLIPSE 100**

Output to the Restart file for ECLIPSE 100 may also be controlled by using the mnemonic RESTART in the RPTSCHED and RPTSOL keywords. However, RPTRST allows greater flexibility when specifying extra data to be written to the Restart file. If the BASIC mnemonic of RPTRST is set to 1 or 2, subsequent use of RPTSCHED overrides the output frequency. However, if BASIC is set to 3 or more, any value of the mnemonic RESTART in RPTSCHED is ignored.

**ECLIPSE 300**

The RESTART mnemonic writes a record from which ECLIPSE 300 can be restarted, using the RESTART keyword in the SOLUTION section.

**ECLIPSE 300**

For back compatibility with the old OUTSOL keyword if no BASIC or RESTART keyword is specified the input quantities are output at every report step until reset.

**ECLIPSE 300**

If this keyword is placed in the SOLUTION section of a flexible restart case, the first record (time=restart time) cannot be used to restart ECLIPSE 300. If a restart is required for this time a Restart file from the original base case should be used.

*Note:* Although the old OUTSOL keyword is superseded, it is still recognized for back compatibility. If a data set contains both OUTSOL and RPTRST, the later one will determine the output to the Restart file.

---

The following table lists the mnemonics available.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACIP</td>
<td>Output pressure-wave impedance (Petro-elastic Model only): this produces an array with the name ACOUIIMPP</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td></td>
<td>If you request this item, you must also request the DEN item.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>These values are only output on the global grid cells (that is, not on local grid refinement cells).</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>ACIS</td>
<td>Output shear-wave impedance (Petro-elastic Model only): this produces an array with the name ACOUIMPS</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>If you request this item, you must also request the DEN item.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>These values are only output on the global grid cells (that is, not on local grid refinement cells).</td>
<td></td>
<td></td>
</tr>
<tr>
<td>AIM</td>
<td>Output AIM status array (1 for explicit cells; 2 for implicit cells)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ALSURF</td>
<td>Output alkaline concentration when surfactant and alkaline models are activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ALSTML</td>
<td>Output alkaline surface tension multiplier when surfactant and alkaline models are activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ALLPROPS</td>
<td>Requests output of the data required by some programs that can interface with ECLIPSE. The output includes fluid densities (OIL_DEN, GAS_DEN and WAT_DEN), viscosities (OIL_VISC, GAS_VISC and WAT_VISC), reciprocal formation volume factors (1/OVERBO, 1/OVERBG and 1/OVERBW) and phase relative permeabilities (OILKR, GASKR and WATKR). If set = 2 the old output mnemonic names (for example 1/FVFOIL instead of 1/OVERBO) are used.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>AMF</td>
<td>Output of aqueous mole fractions (CO2SOL, GASSOL, THERMAL options KWTABTn / CO2STORE and the COMPW option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>AQSP</td>
<td>Output of aqueous speciation (CO2STORE option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>AQPH</td>
<td>Output of aqueous pH level (CO2STORE option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>AREAC</td>
<td>Asphaltene reaction rates</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ASPADS</td>
<td>Asphaltene cumulative adsorption volume fraction (pre-2011.1 model only)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ASPDOT</td>
<td>Asphaltene net cumulative volume fraction deposit (pre-2011.1 model only)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ASPFENT</td>
<td>Asphaltene instantaneous entrainment volume fraction (pre-2011.1 model only)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ASPFLO</td>
<td>Oil velocities (useful to check the asphaltene entrainment) (Asphaltene pre-2011.1 model only)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ASPFLT</td>
<td>Average oil flow rate at the previous timestep (Asphaltene 2011.1 model only)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ASPFRD</td>
<td>Asphaltene precipitate fraction dissolved</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ASPKDM</td>
<td>Asphaltene permeability multiplier (damage) (Asphaltene pre-2011.1 model only)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ASPLIM</td>
<td>Asphaltene precipitate fraction limit</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ASPLUG</td>
<td>Asphaltene instantaneous plugging volume fraction (Asphaltene pre-2011.1 model only)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ASPRET</td>
<td>Asphaltene precipitate fraction (including flocs)</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>
### Keywords

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Description</th>
<th>Symbols</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>ASPREW</td>
<td>Asphaltene precipitate fraction (no flocs)</td>
<td></td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPVEL</td>
<td>Average oil velocity at the previous timestep (Asphaltene 2011.1 model only)</td>
<td></td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPVOM</td>
<td>Asphaltene relative oil viscosity multiplier</td>
<td></td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>BASIC</td>
<td>Output of basic Restart files</td>
<td></td>
<td>x</td>
<td>x</td>
</tr>
</tbody>
</table>
|          | • If set to 1, restart files are created at every report time, but if the files are not unified, only the last one in the run is kept (designed for fail-safe restarts).  
|          | • If set to 2, restart files are created at every report time until this switch is reset, and all are kept.  
|          | • If set to 3, restart files are created every n<sup>th</sup> report time. The frequency of the restarts is governed by the mnemonic "FREQ=n".  
|          | • If set to 4, a Restart file is written at the first report step of each year. Optionally, if the mnemonic FREQ is set >1 the restart is written only every n<sup>th</sup> year.  
|          | • If set to 5, a Restart file is written at the first report step of each month. Optionally, if the mnemonic FREQ is set >1 the restart is written only every n<sup>th</sup> month.  
|          | • If set to 6, a Restart file is written at every timestep.                  |         |      |      |
| Notes:   | If the BASIC=6 mnemonic is used in a base run to generate save or restart information, then it is the timestep number that needs to be specified in item 2 of the LOAD or RESTART keyword when specifying which report step should be restarted from in a fast or flexible restart run using that base run’s restart information. This timestep number can be obtained from the print file of the base run.  
|          | ECLIPSE 300 assumes the case 'BASIC = 2' if the keyword BASIC is used. To subsequently turn off Restarts, use 'BASIC=0'.  
|          | Value 1 is not available with ECLIPSE 300.                                |         |      |      |
| BFORO    | Output of oil Forchheimer B-Factor / phase potential difference (as described in Construction of flows in the ECLIPSE Technical Description). This is only output if VDFLOW, VDFLOWR or VELDEP has been used to activate a non-Darcy flow model.  
|          | Note that this is not the Forchheimer beta parameter.                     |         | x    |      |
| BG       | Gas formation volume factor                                                |         | x    |      |
| BGAS     | Output of gas reservoir molar densities                                     |         | x    |      |
| BIOTCTAB | Geomechanics: outputs Biot’s constant multipliers (stress dependence)      |         | x    |      |
| BO       | Oil formation volume factor                                                |         | x    | x    |
| BOIL     | Output of oil reservoir molar densities                                    |         | x    |      |

---

**ECLIPSE Reference Manual**

1910
<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>BSOL</td>
<td>Output of solid reservoir molar densities (SOLID option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>BTFORG</td>
<td>Output gas Forchheimer beta parameter</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>BTFORO</td>
<td>Output oil Forchheimer beta parameter</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>BW</td>
<td>Water formation volume factor</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>BWAT</td>
<td>Output of water reservoir molar densities</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>CELLINDX</td>
<td>Output the cell indices corresponding to the other output properties. If any cells are missing due to inactivity this allows determination of which cells properties are actually present in the restart file without the need for information in the INIT file.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>The numbering on the global grid is from 1 to NX<em>NY</em>NZ, with the numbering cycling fastest in the X, then Y, then Z directions.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>The numbering on each local grid is from 1 to LX<em>LY</em>LZ, with the numbering cycling fastest in the X, then Y, then Z directions.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>CFL</td>
<td>Outputs CFL numbers (see CFLLIMIT keyword)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>CGAS</td>
<td>Output of grid-block adsorbed gas concentration (for the Coal Bed Methane option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>COHESION</td>
<td>Geomechanics: outputs cohesion (plasticity)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>COLR</td>
<td>Output color of sub-grids (see &quot;The ECLIPSE 300 WARP linear solver&quot; in the of the ECLIPSE Technical Description)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>COILR</td>
<td>Residual oil compressibility (SOR option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>COMPRESS</td>
<td>Requests output of the well connection data on a well by well basis to make the Restart file smaller. This can be useful when a model has large numbers of wells, with just a few wells having a large number of connections.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>CONV</td>
<td>Output of cells which are causing convergence problems. Setting CONV =n focuses on the n worst cells. By default, n=1 (E100) and n=10 (E300). Note:</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>ECLIPSE 100 outputs various convergence problem indicators as described in &quot;Output of cells with convergence problems&quot; in the ECLIPSE Technical Description.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>ECLIPSE 300 outputs various convergence problem indicators as described in &quot;Output of cells with convergence problems&quot; in the ECLIPSE Technical Description.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>DEN</td>
<td>Output of phase reservoir densities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DENG</td>
<td>Output of gas mass densities at reservoir conditions</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DENO</td>
<td>Output of oil mass densities at reservoir conditions</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DENS</td>
<td>Output of solid mass densities (SOLID option) at reservoir conditions</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DEFFSTRS</td>
<td>Delta effective stresses = change from initial conditions: this produces output with the names DEFFSTXX, DEFFSTXY, DEFFSTXZ, DEFFSTYY, DEFFSTYZ, DEFFSTZZ.</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>
### Keywords

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>DENW</td>
<td>Output of water mass densities at reservoir conditions</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DPRSTRES</td>
<td>Delta principal stresses = change from initial conditions; this produces output with the names DPRSTRS1, DPRSTRS2, DPRSTRS3 followed by their respective directions</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DRAIN (AGE)</td>
<td>Output of the drainage regions as DRAINAGE.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DROCKDIS</td>
<td>Geomechanics: outputs Delta rock displacements = change from initial conditions: DROCKDIX, DROCKDIY, DROCKDIZ. These values are interpolated to the centers of the grid blocks.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DSTRAIN</td>
<td>Geomechanics: outputs Delta strain components = change from initial conditions: DSTRANXX, DSTRANXY, DSTRANXZ, DSTRANYY, DSTRANYZ, DSTRANZZ.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DTOTSTRS</td>
<td>Geomechanics: outputs Delta total stresses = change from initial conditions: DTOTSTXX, DTOTSTXY, DTOTSTXZ, DTOTSTYY, DTOTSTYZ, DTOTSTZZ.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>DYNREG</td>
<td>Output of Dynamic Regions</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>EFFSTRES</td>
<td>Geomechanics: outputs rock stresses: EFFSTRXX, EFFSTRXY, EFFSTRXZ, EFFSTRYZ, EFFSTRZZ.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ENERGY</td>
<td>Internal energy / bulk volume (THERMAL option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ESALTS</td>
<td>Effective salinity for the Surfactant Model calculated as specified by the SALTEFF keyword</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ESALTP</td>
<td>Effective salinity for the Polymer Flood Model calculated as specified by the SALTEFF keyword</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>
| FFACTG       | FFACTG = FACTLI \( \times \) \( T_{\text{mix}}^{\text{crit}} / T_{\text{reservoir}} \)  
where \( T_{\text{mix}}^{\text{crit}} \) is defined in the FACTLI keyword,  
Zj used in the FACTLI definition is the gas phase mole fraction |      | x    |
| FFACTO       | FFACTO = FACTLI \( \times \) \( T_{\text{mix}}^{\text{crit}} / T_{\text{reservoir}} \)  
where \( T_{\text{mix}}^{\text{crit}} \) is defined in the FACTLI keyword,  
Zj used in the FACTLI definition is the oil phase mole fraction |      | x    |
| FFORO        | Forchheimer Non-Darcy flow Factor for Oil (VELDEP option)              |      | x    |
### Mnemonic Output

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>FIP</td>
<td>Output of fluids-in-place at separator conditions for ECLIPSE 100, and at reservoir conditions for ECLIPSE 300. For ECLIPSE 100, a basic restart file includes global fluids-in-place only: this mnemonic also outputs local fluids-in-place. Since FIP is simulator-specific it is recommended that the user uses RFIP or SFIP.</td>
</tr>
<tr>
<td>FLOCn</td>
<td>Output of inter-block component flows (for component n, where n is an integer)</td>
</tr>
<tr>
<td>FLOE</td>
<td>Output of inter-block energy flow (ECLIPSE 300 Thermal option)</td>
</tr>
<tr>
<td>FLOGAS</td>
<td>Output of inter-block gas flows</td>
</tr>
<tr>
<td>FLOOIL</td>
<td>Output of inter-block oil flows</td>
</tr>
<tr>
<td>FLOWAT</td>
<td>Output of inter-block water flows</td>
</tr>
<tr>
<td>FLOWS</td>
<td>Output of interblock flows (Includes non-neighbor connection flows and flows between global and local grids.)</td>
</tr>
<tr>
<td>FLORES</td>
<td>Output of interblock flows at reservoir conditions. The volumetric flow rates for each phase are output with headers such as FLRWATI+, FLRWATJ+ and FLRWATK+. Outputs non-neighbor connection flows and flows between global and local grids with headers such as FLRWATN+, FLRWATL+ and FLRWATA+.</td>
</tr>
<tr>
<td>FMISC</td>
<td>Output of relative permeability miscibility factor (Miscible option)</td>
</tr>
<tr>
<td>FOAM</td>
<td>Output of foam surfactant density</td>
</tr>
<tr>
<td>FOAMST</td>
<td>Output of oil-gas surface tension for the Foam Model</td>
</tr>
<tr>
<td>FOAMCNM</td>
<td>Output of capillary number for the Foam Model</td>
</tr>
<tr>
<td>FOAMMOB</td>
<td>Output of gas mobility multiplier for the Foam Model</td>
</tr>
<tr>
<td>FPC</td>
<td>Output of capillary pressure miscibility factor (Miscible option)</td>
</tr>
<tr>
<td>FREQ=n</td>
<td>Controls the frequency of the restart files if BASIC is set &gt;2</td>
</tr>
<tr>
<td>FUGG</td>
<td>Output of component fugacities in gas (Activity Driven Diffusion)</td>
</tr>
<tr>
<td>FUGO</td>
<td>Output of component fugacities in oil (Activity Driven Diffusion)</td>
</tr>
<tr>
<td>GASPOTr</td>
<td>Output of gas potentials (or “depth corrected pressures”)</td>
</tr>
<tr>
<td>GENPLSTN</td>
<td>Geomechanics: outputs generalized plastic strains</td>
</tr>
<tr>
<td>GENPSTRS</td>
<td>Geomechanics: outputs stiffness matrix multiplied by plastic strain.</td>
</tr>
<tr>
<td>GIMULT</td>
<td>Outputs RSGI, RVGI, BOGI and BGGI values</td>
</tr>
<tr>
<td>GOSTRES</td>
<td>Geomechanics: outputs initial rock stresses induced by gravity and overburden: GOESTRXX, GOESTRXY, GOESTRXZ, GOESTRY, GOESTRYZ.</td>
</tr>
<tr>
<td>HGAS</td>
<td>Output gas enthalpy values (ECLIPSE 300 Thermal)</td>
</tr>
<tr>
<td>HOIL</td>
<td>Output oil enthalpy values (ECLIPSE 300 Thermal)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>x</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
</tr>
<tr>
<td>x</td>
<td></td>
</tr>
<tr>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

**Keywords**

1913
<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>HSOL</td>
<td>Output solid enthalpy values (ECLIPSE 300 Thermal and <strong>SOLID</strong> option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>HWAT</td>
<td>Output water enthalpy values (ECLIPSE 300 Thermal)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>HYDH</td>
<td>Output hydraulic head. (Reference data must be supplied in item 1 of the <strong>HYDRHEAD</strong> keyword.)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>HYDHFW</td>
<td>Output freshwater hydraulic head. (Reference data must be supplied in items 1 and 2 of <strong>HYDRHEAD</strong> keyword.)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>JV</td>
<td>Output of ln(K-Value) values</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>KRG</td>
<td>Output of gas relative permeabilities: outputs the array GASKR (E100) or KRG (E300)</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>KRO</td>
<td>Output of oil relative permeabilities: outputs the array OILKR (E100) or KRO (E300)</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>KRW</td>
<td>Output of water relative permeabilities: outputs the array WATKR (E100) or KRW (E300)</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>KRGDM</td>
<td>Output of gas relative permeability damage multiplier</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>KRODM</td>
<td>Output of oil relative permeability damage multiplier</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>KRWDM</td>
<td>Output of water relative permeability damage multiplier</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>LGLCn</td>
<td>Output of number of <strong>LAGTRAC</strong> streamlines generated for component n to pass through a cell</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>LGLCWAT</td>
<td>Output of number of <strong>LAGTRAC</strong> streamlines generated for water to pass through a cell</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>LGLCHC</td>
<td>Output of number of <strong>LAGTRAC</strong> streamlines generated for hydrocarbons (that is to solve <strong>LUMPING</strong> tracers) to pass through a cell</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MLSC</td>
<td>Output of total molar density for hydrocarbon components (moles per reservoir volume)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MWAT</td>
<td>Output of total molar density for water components (moles per reservoir volume)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>NCNG</td>
<td>Output of capillary numbers for gas (<strong>VELDEP</strong> option). The normalized capillary number is output when used with keyword <strong>VDKRG</strong>.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>NCNO</td>
<td>Output of capillary numbers for oil (<strong>VELDEP</strong> option). The normalized capillary number is output when used with keyword <strong>VDKRO</strong>.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>NOGRAD</td>
<td>Output of total derivatives for the Gradient option</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

If set > 0, extra information is not written to the Restart file for the Gradient option.
### Mnemonic: NORST

Requests a graphics only Restart file.

- If set to 1, the Restart file does not contain arrays required for restarting but not usually required for graphical output (for example hysteresis arrays).
- If set to 2, the well arrays are not included on the restart file. In a standard black oil case the restart file will contain header data and 4 arrays, PRESSURE, SWAT, SGAS and RS (and any other arrays requested by the other controls in this keyword).

The restart files dumped while this control is set > 0 cannot be used to restart the run. You could, however, opt to have the restart data written to the Save file (see the mnemonic SAVE, and keyword RESTART).

### Mnemonic: NPMREB

Output of the number of passes through the rebound region during the calculations of the pore volumes of the Palmer-Mansoori rock model (keyword ROCKPAMA or ROCKPAME). Also gives the sign of the derivatives during the last calculation (positive >=0). Requires CBMOPTS item 4 to be set to PMREB.

### Mnemonic: OILPOT

Output of oil potentials (or “depth corrected pressures”)

### Mnemonic: PART

Output of domain partitions (Parallel option)

### Mnemonic: PBPD

Output of bubble point and dew point pressures

### Mnemonic: PCGW

Output of gas-water capillary pressures (GASWAT cases)

### Mnemonic: PCOG

Output of gas-oil capillary pressures

### Mnemonic: PCOW

Output of oil-water capillary pressures

### Mnemonic: PERMSTAB

Geomechanics: outputs permeability multipliers (stress dependence)

### Mnemonic: PERM_MDX, PERM_MDY, PERM_MDZ

Output directional permeability multipliers (Rock Compaction option) when RKTRMDIR or ROCKTRMX, ROCKTRMY and ROCKTRMZ are defined in ECLIPSE 300.

This information can be output in ECLIPSE 100 via the ROCKC mnemonic (described below).

### Mnemonic: PERM_MOD

Output permeability multipliers (Rock Compaction option).

This information can be output in E100 via the ROCKC mnemonic (described below).

### Mnemonic: PGAS

Output of gas phase pressure

### Mnemonic: PGOSTRES

Geomechanics: outputs initial principal stresses induced by gravity and overburden: GVPRSTS1, GVPRSTS2, and GVPRSTS3.

### Mnemonic: PKRG

Processed gas relative permeability (ENKRVC and ENKRVT cases)

### Mnemonic: PKRGR

Output of the processed scaled gas relative permeability at the residual oil saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the ENKRVC or ENKRVT keywords respectively.

### Mnemonic: PKRO

Processed oil relative permeability (ENKRVC and ENKRVT cases)
<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>PKRORG</td>
<td>Output of the processed scaled oil relative permeability at the residual gas saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the <code>ENKRVC</code> and <code>ENKRVT</code> keywords respectively.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PKRORW</td>
<td>Output of the processed scaled oil relative permeability at the residual water saturation for each cell if composition or temperature-dependent end-point scaling is selected using the <code>ENKRVC</code> or <code>ENKRVT</code> keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PKRW</td>
<td>Processed water relative permeability (<code>ENKRVC</code> and <code>ENKRVT</code> cases)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PKRW</td>
<td>Output of the processed scaled water relative permeability at the residual oil saturation for each cell if composition or temperature-dependent end-point scaling is selected using the <code>ENKRVC</code> or <code>ENKRVT</code> keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PLASDISP</td>
<td>Geomechanics: outputs plastic displacements</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>POIL</td>
<td>Output of oil phase pressure</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>POIS</td>
<td>Output Poisson’s ratio (Petro-elastic Model only): this produces an array with the name <code>POISSONR</code> If you request this item, you must also request the <code>DEN</code> item These values are only output on the global grid cells (that is, not on local grid refinement cells).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>POLY</td>
<td>Output of polymer concentration (Polymer option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>POLYVM</td>
<td>Output of polymer viscosity multiplier (Polymer option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PORO</td>
<td>Output of effective porosities (Gradient Option only)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PORV</td>
<td>Output of reservoir pore volumes at reference conditions</td>
<td></td>
<td></td>
</tr>
<tr>
<td>POT</td>
<td>Output of phase potentials (or “depth corrected pressures”): this produces the arrays <code>OIL_POTN</code>, <code>GAS_POTN</code> and <code>WAT_POTN</code>. (Including the initial contact corrected potential output to the array <code>POT_CORR</code>.)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PORV_MOD</td>
<td>Output pore volume multipliers (Rock compaction option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PPCG</td>
<td>Output of processed capillary pressure gas-oil end-points</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PPCW</td>
<td>Output of processed capillary pressure oil-water end-points</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PRES_EFF</td>
<td>Output of effective fluid pressures for rock compaction (when <code>OVERBURD</code> and <code>ROCKOPTS</code> are specified).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PRES</td>
<td>ECLIPSE 100 Output of water and gas phase pressures ECLIPSE 300 Output of grid block pressures</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PRESMIN</td>
<td>Output of minimum pressures (Irreversible rock compaction option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PRESSURE</td>
<td>Output of grid block pressures (equivalent to <code>PRES</code>)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>--------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>PRSTRESS</td>
<td>Geomechanics: outputs principal stresses: PRSTRES1, PRSTRES2, PRSTRES3 followed by their respective directions.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSAT</td>
<td>Output of saturation pressures (Compositional mode). Note: Saturation pressures (PSAT) are not normally used in ECLIPSE 300, and their calculation may involve significant extra CPU time. The value of PSAT depends on the cell hydrocarbon state: for single phase gas it will usually be the dew point pressure; for single phase oil it will usually be the bubble point pressure; for two phase cells it will be the cell pressure. In thermal simulations, the value of PSAT, for both rock-filled cells and for water-filled cells, is the water saturation pressure at the cell temperature.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSGCR</td>
<td>Output of the processed scaled critical gas saturation end-points for each grid cell if composition or temperature-dependent end-point scaling is selected via the use of the ENPTVC or ENPTVT keywords respectively or the mobile fluid correction options or TZONE are activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSGL</td>
<td>Output of the processed scaled minimum gas saturation end-points for each grid cell if composition or temperature-dependent end-point scaling is selected using the ENPTVC or ENPTVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSGU</td>
<td>Output of the processed scaled maximum gas saturation end-points for each grid cell if composition or temperature-dependent end-point scaling is selected using the ENPTVC or ENPTVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSOGCR</td>
<td>Output of the processed scaled critical oil-in-gas saturations for each grid cell if composition or temperature-dependent end-point scaling is selected using the ENPTVC or ENPTVT keywords respectively or the mobile fluid correction options or TZONE are activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSOWCR</td>
<td>Output of the processed scaled critical oil-in-water saturations for each grid cell if composition or temperature-dependent end-point scaling is selected using the ENPTVC or ENPTVT keywords respectively or the mobile fluid correction options or TZONE are activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSWCR</td>
<td>Output of the processed scaled critical water saturation end-points for each grid cell if composition or temperature-dependent end-point scaling is selected using the ENPTVC or ENPTVT keywords respectively or the mobile fluid correction options or TZONE are activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSWL</td>
<td>Output of the processed scaled minimum water saturation end-points for each grid cell if composition or temperature-dependent end-point scaling is selected using the ENPTVC or ENPTVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSWU</td>
<td>Output of the processed scaled maximum water saturation end-points for each grid cell if composition or temperature-dependent end-point scaling is selected via the use of the ENPTVC or ENPTVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PTCSTRES</td>
<td>Geomechanics: outputs initial principal stresses induced by tectonic loadings: TCPRSTS1, TCPRSTS2 and TCPRSTS3.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PVDPH</td>
<td>Derivative of pore volume with respect to porosity</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PWAT</td>
<td>Output of water phase pressure</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RATP</td>
<td>Ratio of actual to target throughputs</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>RATS</td>
<td>Ratio of actual to target solution changes</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RATT</td>
<td>Ratio of actual to target temperature changes (Thermal option only)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>REAC</td>
<td>Chemical reaction rates (See &quot;Chemical Reactions&quot; in the <em>ECLIPSE Technical Description</em>)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RESTART</td>
<td>Output of data required for flexible restart</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RFIP</td>
<td>Output of fluids-in-place at reservoir conditions. In ECLIPSE 100 this is the product of pore volume at reservoir conditions with saturation. In ECLIPSE 300, this is a synonym for FIP.</td>
<td></td>
<td>x x</td>
</tr>
<tr>
<td>RK or PERMREDN</td>
<td>Polymer Flood: Permeability reduction for the aqueous phase due to polymer retention in the rock.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ROCKC</td>
<td>Output of pore volume and permeability modifiers for the Rock Compaction option. In ECLIPSE 300, this is the same as individually specifying mnemonics PORV_MOD, PERM_MOD (or PERM_MDX, PERM_MDY and PERM_MDZ in the directional case) and optionally SIGM_MOD (in a dual porosity model). In ECLIPSE 100, the values output (where relevant) are PRES_OVB, PRES_EFF, PEFF_MIN, PORV_MOD, PERM_MOD (or PERM_MDX, PERM_MDY and PERM_MDZ in the directional case).</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>ROCKDISP</td>
<td>Geomechanics: outputs rock displacements: ROCKDISX, ROCKDISY, ROCKDISZ. These values are interpolated to the centers of the grid blocks.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ROMLS</td>
<td>Residual oil composition (SOR option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RPORV</td>
<td>Output of reservoir pore volumes at Reservoir conditions</td>
<td></td>
<td>x x</td>
</tr>
<tr>
<td>RS</td>
<td>Output dissolved GOR (R_s) (In compositional runs this is interpreted as the GOR at surface conditions of the liquid phase in the reservoir)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RSSAT</td>
<td>Output of saturated R_s values (dissolved gas-oil ratio). Only in Blackoil mode for ECLIPSE 300.</td>
<td></td>
<td>x x</td>
</tr>
<tr>
<td>RSW</td>
<td>Output of the aqueous concentration of soluble components. Each component is distinguished, as with mnemonic AMF, by its associated index.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RV</td>
<td>Output vaporized OGR (R_v) (In compositional runs this is interpreted as the OGR at surface conditions of what was the vapor phase in the reservoir.)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RVSAT</td>
<td>Output of saturated R_v values (vaporized oil-gas ratio). Only in Blackoil mode for ECLIPSE 300.</td>
<td></td>
<td>x x</td>
</tr>
</tbody>
</table>
### Mnemonic | Output | E100 | E300
--- | --- | --- | ---
SAVE | Requests a Restart file be written to the Save file. This item has a similar effect to item 1, the BASIC restart control, but governs the writing of time-dependent restart data to the Save file.  
- If set to 1 or 2, restart data is written to the Save file every report time until this switch is reset.  
- If set to 3, restart data is written every n<sup>th</sup> report time. The frequency of the restarts is governed by the mnemonic.  
- If set to 4, restart data is written at the first report step of each year. Optionally, if SFREQ is set >1 the restart is written only every n<sup>th</sup> year.  
- If set to 5, restart data is written at the first report step of each month. Optionally, if SFREQ is set >1 the restart is written only every n<sup>th</sup> month.  
- If set to 6, this setting is reserved for use by FrontSim.  
Note that the Save file must be requested either by using the SAVE keyword in the RUNSPEC section or by requesting a Save file in the LOAD keyword, otherwise no restart data is output to the Save file. | x | |
SDENO | Output of grid block oil surface densities (for API tracking) | x | |
SFIP | Output of fluids-in-place at surface/separator conditions (see "Initial fluid in place" and item 201 of OPTIONS3) | x | x
SFIPGAS | Output of separator volumes for gas FIP data | x | |
SFIPOIL | Output of separator volumes for oil FIP data | x | |
SFIPWAT | Output of separator volumes for water FIP data (single component water only at present) | x | |
SFOIL | Output of grid block foamy oil saturation | x | |
SFREQ=n | Controls the frequency of the restart files written to the Save file if the mnemonic SAVE is set >2 | x | |
SFSOL | Output of grid block foamy solid saturation | x | |
SGAS | Output of grid block gas saturations | x | |
SGASMAX | Output of maximum gas saturation (Hysteresis option) | x | |
SGCRH | Output critical gas saturation (Hysteresis option) | x | |
SGTRH | Output the immobile gas saturation (Hysteresis option) | x | |
SGTRAP | Output of the trapped gas saturation $S_{\text{grap}}$ for the WAG hysteresis option | x | x
SIGM_MOD | Output dual porosity sigma multipliers (Rock Compaction option) | x | x
SMF | Output of solid component mole fractions (CO2STORE option only) | x | |
SMMULT | Output of solid mobility multiplier (SOLID option) | x | |
SOIL | Output of grid block oil saturations | x | |
SOILM | Mobile oil saturation (SOR option) | x | |

**Keywords**

1919
<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>SOILMAX</td>
<td>Output of maximum oil saturation (Hysteresis option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOILR</td>
<td>Residual oil saturation (SOR option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOLADS</td>
<td>Adsorbed solid concentration (Solid option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOLADW</td>
<td>Adsorbed wetting-solid concentration (Solid option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOLWET</td>
<td>Solid wettability interpolation parameter (Solid option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SSFRAC</td>
<td>Output of suspended solids fraction (Solid option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SSOLID</td>
<td>Output of grid block solid saturations (Solid option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>STATE</td>
<td>Output of states present</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>STEN</td>
<td>Output of oil/gas surface tensions (Miscible option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>STRAIN</td>
<td>Geomechanics: outputs strain components: STRAINXX, STRAINXY, STRAINXZ, STRAINYY, STRAINYZ, STRAINZZ. Note that these do not form a tensor - see discussion on definition of strains in “Elastic Stress Equations” of the ECLIPSE Technical Description.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>STREAM=n</td>
<td>Reserved for use by FrontSim</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SUBG</td>
<td>Output the sub-grid (see &quot;The ECLIPSE 300 WARP linear solver&quot; in the ECLIPSE Technical Description)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SURF</td>
<td>Surfactant concentration - Surfactant option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SURFBLK</td>
<td>When NORST mnemonic is activated, output of surfactant concentrations (SURFACT), maximum surfactant concentrations (SURFMAX), adsorption (SURFADS), log (capillary number) (SURFCNM) and water oil surface tension (SURFST or SURFSTES). Surfactant option.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SURFCNM</td>
<td>log (capillary number) - Surfactant option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SURFKR</td>
<td>Relative permeability interpolation parameter - Surfactant option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SURFCP</td>
<td>Capillary pressure multiplier - Surfactant option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SURFST</td>
<td>Oil-water surface tension - Surfactant option</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SWAT</td>
<td>Output of grid block water saturations</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SWATMIN</td>
<td>Output of minimum water saturation (Hysteresis option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>TCBULK</td>
<td>Output of the average thermal conductivity (using the mnemonic THCAVE specified in the RPTGRID keyword) multiplied by saturation dependent thermal transmissibility multiplier ( A ) (THERMAL option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>TCMULT</td>
<td>Saturation dependent thermal transmissibility multiplier ( A ) (THERMAL option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>TCSTRES</td>
<td>Geomechanics: outputs initial rock stresses induced by tectonic loadings: TCESTRXX, TCESTRXY, TCESTRXZ, TCESTRYY, TCESTRYZ, TCESTRZZ</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>TEMP</td>
<td>Output of reservoir temperatures (THERMAL option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>TOTCOMP</td>
<td>Outputs total compressibility of the fluid system (excluding the rock compressibility)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>------------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>TOTSTRES</td>
<td>Geomechanics: outputs total stresses: TOTSTRXX, TOTSTRXY, TOTSTRXZ, TOTSTRYY, TOTSTRYZ, TOTSTRZZ.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TRAS</td>
<td>Output the product of tracer concentration with the carrier saturation</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>TREACM</td>
<td>Tracer dependent reaction rate multipliers (See &quot;Chemical Reactions&quot; in the ECLIPSE Technical Description)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TRMFxxxx</td>
<td>Output the product of tracer concentration with the molar fraction of the carrier (molar fraction relative to all components, not just hydrocarbons). To output use TRMF followed by the tracer name.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TSUB</td>
<td>Output of the water saturation temperature minus the grid block temperature (THERMAL option)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VELGAS</td>
<td>Output of gas velocities</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>This gives phase velocity $u_p$ as defined in &quot;ECLIPSE 100 Velocity calculations&quot; in the ECLIPSE Technical Description.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VELOCITY</td>
<td>Output of all fluid velocities.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>This gives phase velocity $u_p$ as defined in &quot;ECLIPSE 100 Velocity calculations&quot; in the ECLIPSE Technical Description.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>For polymer shear-thinning in the Polymer Flood Model, only water velocity is output.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>For shear thinning in the Foam Model, only gas velocity is output.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VEOIL</td>
<td>Output of oil velocities</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>This gives phase velocity $u_p$ as defined in &quot;ECLIPSE 100 Velocity calculations&quot; in the ECLIPSE Technical Description.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VELWAT</td>
<td>Output of water velocities</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>This gives phase velocity $u_p$ as defined in &quot;ECLIPSE 100 Velocity calculations&quot; in the ECLIPSE Technical Description.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VGAS</td>
<td>Output of gas viscosities</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>VISC</td>
<td>Argument VISC allows to output extra viscosities when Polymer is active. You can obtain the following output:</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• EPVIS: effective polymer viscosity (Polymer option).</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• EMVIS: effective mixture (water/polymer) viscosity (Polymer option).</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• EWV_POL: effective water viscosity (Polymer option).</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• EWV_SAL: effective water viscosity due to salt concentration (Brine option).</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• PLADALK: polymer adsorption multiplier as a function of alkaline concentration.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• SFIPPLY/SFIPPOL: polymer in place (mass)</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• RRFFAC: residual resistance factor.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• TRXBYRRF: I-direction transmissibility divided by the residual resistance factor.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• TRYBYRRF: J-direction transmissibility divided by the residual resistance factor.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• TRZBYRRF: K-direction transmissibility divided by the residual resistance factor.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• SHWVISI, SHWVISJ, SHWVISK: shear viscosity of the water/polymer solution due to shear thinning/thickening in the positive I-, J- and K-directions, respectively (PLYSHEAR and PLYSHLOG keywords).</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• POLYMAX: maximum polymer concentration.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• RRFTMAX: RRF maximum temperature.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• POLYMER: polymer concentration.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VOIL</td>
<td>Output of oil viscosities</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VMF</td>
<td>Output of vapor mole fractions (fraction of total hydrocarbon moles in gas)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VWAT</td>
<td>Output of water viscosities</td>
<td></td>
<td></td>
</tr>
<tr>
<td>WATPOT</td>
<td>Output of water potentials (or “depth corrected pressures”)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>XFW</td>
<td>Output of liquid component mass fractions (weight fraction of each component in liquid)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>XGAS</td>
<td>Output of gas fractional saturations (Sudation option)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>XMF</td>
<td>Output of liquid component mole fractions (fraction of each component in liquid)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>XWAT</td>
<td>Output of water fractional saturations (Sudation option)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>YFW</td>
<td>Output of vapor component mass fractions (weight fraction of each component in gas)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
YMF

Output of vapor component mole fractions (fraction of each component in gas)

ZMF

Output of total component mole fractions

Table 3.37: RPTRST output controls

The following table lists the deprecated mnemonics.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>PKRGE</td>
<td>This has been superseded by the mnemonic PKRGR (ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PKROGE</td>
<td>This has been superseded by the mnemonic PKROGR (ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PKROWE</td>
<td>This has been superseded by the mnemonic PKRORW (ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PKRWE</td>
<td>This has been superseded by the mnemonic PKRWR (ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGCR</td>
<td>This has been superseded by the mnemonic PSGCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGL</td>
<td>This has been superseded by the mnemonic PSGL (ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGU</td>
<td>This has been superseded by the mnemonic PSGU (ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOGCR</td>
<td>This has been superseded by the mnemonic PSOGCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOWCR</td>
<td>This has been superseded by the mnemonic PSOWCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWCR</td>
<td>This has been superseded by the mnemonic PSWCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWL</td>
<td>This has been superseded by the mnemonic PSWL (ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWU</td>
<td>This has been superseded by the mnemonic PSWU (ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>WMF</td>
<td>The mnemonic has been superseded by AMF.</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

Table 3.38: Deprecated RPTRST output controls

XMF/YMF output

If either XMF or YMF is requested, a set of values is written to the Restart file for each component. If for instance there are 5 components and the XMF mnemonic is used, then ECLIPSE 300 writes mnemonics XMF1, XMF2, XMF3, XMF4 and XMF5 to the restart file. These can be displayed in GRAF by asking for XMF1 and so on.

Saturation pressures are not normally used in ECLIPSE 300, and their calculation may involve significant extra CPU time.
Tracer output

**ECLIPSE 300**

To output the distribution of a tracer, add the name of the tracer as one of the mnemonics.

**ECLIPSE 100**

ECLIPSE 100 always outputs the tracer distributions. This will be the tracer name followed by F (Free) and S (Solution) specifiers, or by the phase number # in the case of partitioned tracers, as appropriate.

Output of flows

**ECLIPSE 100**

Under the FLOWS mnemonic, it is possible to output the interblock flows, including non-neighbor connection flows, and flows between global and local grids. For instance, the flow between neighbor cells in the water phase will be defined under the FLOWATI+, FLOWATJ+, FLOWATK+ headers for I, J, K positive directions. Flow between non-neighbor connections can be found under the FLOWATN+ header. The actual non-neighbor connections can be found in the grid file.

**ECLIPSE 300**

RPTRST/OUTSOL items exist that enable flows to be written to the Restart files. These may be used to produce arrow diagrams showing fluid movement. The mnemonics are FLOC1 for component 1, FLOC2 for component 2 and so on (as molar rates). FLOWAT outputs the water flow rate (as a surface rate). FLOOIL and FLOGAS output the surface flow rates of oil and gas, respectively. In thermal simulations, FLOE outputs the energy flow rate.

Where appropriate, interblock flows will be output to the restart files using a similar naming convention as for ECLIPSE 100, giving data with headers such as FLOC31+ or FLOC1N+ in the restart file.

Extra output for BASIC=2

**ECLIPSE 100**

When BASIC=2 is used in ECLIPSE 100, additional output is provided. For instance

<table>
<thead>
<tr>
<th>BASIC=2</th>
<th>Additional Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>PRESSURE</td>
<td>Output of grid block pressures</td>
</tr>
<tr>
<td>RS</td>
<td>Output of dissolved GOR (Rs)</td>
</tr>
<tr>
<td>RV</td>
<td>Output of vaporized OGR (Rv)</td>
</tr>
<tr>
<td>SWAT</td>
<td>Output of grid block water saturations</td>
</tr>
<tr>
<td>SGAS</td>
<td>Output of grid block gas saturations</td>
</tr>
<tr>
<td>OILAPI</td>
<td>Output of grid block API values (for API tracking)</td>
</tr>
</tbody>
</table>

*Table 3.39: Extra output for BASIC=2 in ECLIPSE 100*

**ECLIPSE 300**

When BASIC=2 is used in ECLIPSE 300, additional output is provided. For instance,

<table>
<thead>
<tr>
<th>BASIC=2</th>
<th>Additional Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>Asphaltene 2011.1 model</td>
<td>ASPFLT</td>
</tr>
<tr>
<td>Asphaltene 2011.1 model</td>
<td>ASPVEL</td>
</tr>
<tr>
<td>Residual oil model</td>
<td>COILR</td>
</tr>
<tr>
<td>Compositional models</td>
<td>LIFACT</td>
</tr>
<tr>
<td>MLSC $1,2,\ldots,n$</td>
<td>Hydrocarbon moles/reservoir volume for each component, $1,2,\ldots,n$ (&quot;total molar density&quot;)</td>
</tr>
<tr>
<td>MWAT</td>
<td>Water moles/reservoir volume for each water component (&quot;total molar density of water&quot;)</td>
</tr>
</tbody>
</table>
In addition, more well information is also provided, such as mixture data, completion data, separator data and group data.

Some data, such as $SGT$ and $SWT$, are only output if the $DPCDT$ option is on. Note that this can be on without explicitly having the keyword present.

When BASIC=2 is used in ECLIPSE 300 with the Thermal Option and in combination with $ENDSCALE$ item 5 set to 1 or 2, the following set of reference grid-block end-points are output. These correspond to the grid-block end-points defined by the input end-point scaling arrays and are used in conjunction with the temperature-dependent tables to construct the temperature-dependent end-points.

<table>
<thead>
<tr>
<th>BASIC=2</th>
<th>Additional output</th>
</tr>
</thead>
<tbody>
<tr>
<td>$RKRGR$</td>
<td>Reference scaled maximum gas relative permeability for each cell.</td>
</tr>
<tr>
<td>$RKRGR$</td>
<td>Reference scaled gas relative permeability at the residual oil saturation for each cell.</td>
</tr>
<tr>
<td>$RKRO$</td>
<td>Reference scaled maximum oil relative permeability for each cell.</td>
</tr>
<tr>
<td>$RKROG$</td>
<td>Reference scaled oil relative permeability at the residual gas saturation for each cell.</td>
</tr>
<tr>
<td>$RKRORW$</td>
<td>Reference scaled oil relative permeability at the residual water saturation for each cell.</td>
</tr>
<tr>
<td>$RKRW$</td>
<td>Reference scaled maximum water relative permeability.</td>
</tr>
<tr>
<td>$RKRW$</td>
<td>Reference scaled water relative permeability at the residual oil saturation for each cell.</td>
</tr>
<tr>
<td>$RPCG$</td>
<td>Reference scaled maximum gas-oil capillary pressure for each cell.</td>
</tr>
<tr>
<td>$RPCW$</td>
<td>Reference scaled maximum water-oil capillary pressure for each cell.</td>
</tr>
</tbody>
</table>

Table 3.40: Extra output for BASIC=2 in ECLIPSE 300
Table 3.41: Extra output for BASIC=2 in ECLIPSE 300 for processed end-points.

<table>
<thead>
<tr>
<th>BASIC=2</th>
<th>Additional output</th>
</tr>
</thead>
<tbody>
<tr>
<td>RSOGCR</td>
<td>Reference scaled critical oil-in-gas saturation for each grid cell.</td>
</tr>
<tr>
<td>RSOWCR</td>
<td>Reference scaled critical oil-in-water saturation for each grid cell.</td>
</tr>
<tr>
<td>RSGCR</td>
<td>Reference scaled critical gas saturation for each grid cell.</td>
</tr>
<tr>
<td>RSGL</td>
<td>Reference scaled minimum gas saturation for each grid cell.</td>
</tr>
<tr>
<td>RSGU</td>
<td>Reference scaled maximum gas saturation for each grid cell.</td>
</tr>
<tr>
<td>RSWCR</td>
<td>Reference scaled critical water saturation for each grid cell.</td>
</tr>
<tr>
<td>RSWL</td>
<td>Reference scaled minimum water saturation for each grid cell.</td>
</tr>
<tr>
<td>RSWU</td>
<td>Reference scaled maximum water saturation for each grid cell.</td>
</tr>
</tbody>
</table>

Examples

Example 1 (ECLIPSE 100)

```
RPTRST
BASIC=2 FLOWS POT PBPD /
```

Example 2

In ECLIPSE 300 it is not possible to restart the simulator with this example but PRESSURE, SOIL and VGAS are output to every Restart file until reset.

```
RPTRST
PRESSURE SOIL VGAS /
```
RPRUNSP  Controls output of RUNSPEC data

If this keyword is specified in the RUNSPEC section, a listing of the parameters set in the RUNSPEC section will be written to the Print file. This output includes all variables set in the RUNSPEC section (both default and user defined values).

The keyword has no associated data.

Keywords

x  |  ECLIPSE 100  
   |  ECLIPSE 300  
   |  SPECIAL  

x  |  RUNSPEC  
   |  GRID  
   |  EDIT  
   |  PROPS  
   |  REGIONS  
   |  SOLUTION  
   |  SUMMARY  
   |  SCHEDULE  

1927
**RPTSCHED**  Controls on output from SCHEDULE section

The keyword should be followed by a list of mnemonics which control the output of SCHEDULE section data to the Print file. Wherever possible, the mnemonics correspond to the related keywords. The list should be terminated by a slash (/).

The RPTSCHED keyword may be used to reset output controls for the SCHEDULE section as often as required.

Older ECLIPSE 100 data sets may not use mnemonics but integer controls. The meaning of these is documented in "RPTSCHED" in the ECLIPSE Technical Description.

**Note:** Do not use these integer controls in ECLIPSE 300.

In ECLIPSE 100

- Parallel runs will not output all requested quantities for performance reasons. The output can be enabled by setting item 8 of PARAOPTS to a value greater than zero.
- Entering the mnemonic NOTHING clears all the reporting flags.
- For additional control over the output, integer values greater than one may be assigned to selected mnemonics using the syntax:

  mnemonic=integer

  where there must be no spaces either side of the equals sign.
- By default, the printed tables of values are rescaled so that the values printed are have significant figures within the decimal digits printed; however not all scalings are allowed and so not all significant digits may be printed. If insufficient digits are printed, then OPTIONS 235 may be set to 1. With this scaling option, the maximum value is always between 1 and 10 and all its significant digits will be printed; all possible scalings in the valid range for floating point variables are available.

The following table lists the mnemonics available. Mnemonics are identified by an x in the appropriate column.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>AIM</td>
<td>Output of AIM status array (1 for implicit cells; 2 for explicit cells).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ALKALINE</td>
<td>Output of alkaline concentrations and alkaline adsorption when activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ALSURF</td>
<td>Output alkaline concentration when surfactant and alkaline models are activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ALSTML</td>
<td>Output alkaline surface tension multiplier when surfactant and alkaline models are activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>AMP</td>
<td>Output of aqueous mole fractions (CO2SOL, GASSOL and THERMAL options KWATBTN / CO2STORE and the COMFW option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ANIONS</td>
<td>Output of anion concentration.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>AQSP</td>
<td>Output of aqueous speciation (CO2STORE option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>AQPHT</td>
<td>Output of aqueous pH level (CO2STORE option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>--------------</td>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>AQUCT or AQUFET or AQUFETP</td>
<td>Output of Fetkovich or Carter-Tracy aquifer status. Also outputs AUCHWAT, AUCHGAS and AQUFLUX status. Note: AQUFET applies only for ECLIPSE 100.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>AREAC</td>
<td>Asphaltene reaction rates.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPADS</td>
<td>Asphaltene cumulative adsorption volume fraction (pre-2011.1 model only).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPDOT</td>
<td>Asphaltene net cumulative volume fraction deposit (pre-2011.1 model only).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPFENT</td>
<td>Asphaltene instantaneous entrainment volume fraction (pre-2011.1 model only).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPFLO</td>
<td>Oil velocities (useful to check the asphaltene entrainment) (pre-2011.1 model only).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPFLT</td>
<td>Average oil flow rate at the previous timestep (2011.1 model only).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPFRD</td>
<td>Asphaltene precipitate fraction dissolved.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPKDM</td>
<td>Asphaltene permeability multiplier (damage) (pre-2011.1 model only).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPLIM</td>
<td>Asphaltene precipitate fraction limit.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPUG</td>
<td>Asphaltene instantaneous plugging volume fraction (pre-2011.1 model only).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPRET</td>
<td>Asphaltene precipitate fraction (including flocs).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPREW</td>
<td>Asphaltene precipitate fraction (no flocs).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPVEL</td>
<td>Average oil velocity at the previous timestep (2011.1 model only).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ASPVOM</td>
<td>Asphaltene relative oil viscosity multiplier.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>BFORG</td>
<td>Output of gas Forchheimer B-Factor / phase potential difference (as described in Construction of flows in the ECLIPSE Technical Description). This is only output if VDFLOW or VDFLOWR (or VELDEP in ECLIPSE 300) has been used to activate a non-Darcy flow model. Note that this is not the Forchheimer beta parameter.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>BFORO</td>
<td>Output of oil Forchheimer B-Factor / phase potential difference (as described in Construction of flows in the ECLIPSE Technical Description). This is only output if VDFLOW, VDFLOWR or VELDEP has been used to activate a non-Darcy flow model. Note that this is not the Forchheimer beta parameter.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>BGAS</td>
<td>Output of gas reservoir molar densities.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>BIOTCTAB</td>
<td>Geomechanics: outputs Biot’s constant multipliers (stress dependence).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>BOIL</td>
<td>Output of oil reservoir molar densities.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>BSOL</td>
<td>Output of solid reservoir molar densities (SOLID option).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>BTFORG</td>
<td>Output of gas Forchheimer beta parameter.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>BTFORO</td>
<td>Output of oil Forchheimer beta parameter.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>BWAT</td>
<td>Output of water reservoir molar densities.</td>
<td>x</td>
<td></td>
</tr>
</tbody>
</table>
### Mnemonic | Output | E100 | E300
--- | --- | --- | ---
**CATIONS** | Output of divalent cation concentration. | x |  
**CGAS** | Output of grid-block adsorbed gas concentration (for the Coal Bed Methane option). | x |  
**COHESION** | Geomechanics: outputs cohesion (plasticity). | x |  
**COILR** | Residual oil compressibility (SOR option). | x |  
**CPU** | Output of breakdown of CPU and elapsed time for current run.  
1. report is sent to Print file.  
2. in addition to being sent to the Print file, a copy of the report is sent to the short form output (terminal)  
3. report is sent to short form output only.  
4. as 2 but with actual cpu and elapsed times output in seconds.  
See “Timing Reports” in the ECLIPSE Technical Description for an explanation of the outputs. Internal clocks within TS are not recorded during intermediate timesteps when set to 0. | x | x  
**DENG** | Output of grid block gas reservoir density. | x | x  
**DENO** | Output of grid block oil reservoir density. | x | x  
**DENS** | Output of grid block solid reservoir density (SOLID option). | x |  
**DEFFSTRS** | Delta effective stresses = change from initial conditions: this produces output with the names DEFFSTXX, DEFFSTXY, DEFFSTXZ, DEFFSTYY, DEFFSTYZ, DEFFSTZZ. | x |  
**DENW** | Output of grid block water reservoir density. | x | x  
**DPRSTRES** | Delta principal stresses = change from initial conditions: this produces output with the names DPRSTRS1, DPRSTRS2, DPRSTRS3 followed by their respective directions. | x |  
**DROCKDIS** | Geomechanics: outputs Delta rock displacements = change from initial conditions: DROCKDIX, DROCKDIY, DROCKDIZ. These values are defined at the faces of the grids. | x |  
**DSTRAIN** | Geomechanics: outputs Delta strain components = change from initial conditions: DSTRANXX, DSTRANXY, DSTRANXZ, DSTRANYY, DSTRANYZ, DSTRANZZ. | x |  
**DTOTSTRS** | Geomechanics: outputs Delta total stresses = change from initial conditions: DTOTSTXX, DTOTSTXY, DTOTSTXZ, DTOTSTYY, DTOTSTYZ, DTOTSTZZ. | x |  
**EFFSTRES** | Geomechanics: outputs rock stresses: EFFSTRXX, EFFSTRXY, EFFSTRXZ, EFFSTRYY, EFFSTRYZ, EFFSTRZZ. | x |  
**ENERGY** | Output total energies/bulk volume (energy/(rock+pore volume)). ECLIPSE 300 Thermal. | x |  
**ESALTP** | Output of effective salinity for polymer properties. | x |  
**ESALPLY** | Output of effective salinity for polymer properties. | x |  
**ESALTS** | Output of effective salinity for phase behavior and surfactant properties. | x |  

**Keywords**

1930
<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>ESALSUR</td>
<td>Output of effective salinity for phase behavior and surfactant properties.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FFORG</td>
<td>ECLIPSE 300: output of Forchheimer Non-Darcy flow factor for Gas (VELDEP option). ECLIPSE 100: Maximum deviation from Darcy Flow.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>FFORO</td>
<td>Output of Forchheimer Non-Darcy flow factor for Oil (VELDEP option). ECLIPSE 100: not applicable yet.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FIP</td>
<td>Output of fluid in place and inter-region flow reports:</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>1. fluids in place are reported for the whole field.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2. in addition, a balance sheet is produced for each fluid in place region defined with the FIPNUM keyword. This shows current and initial fluids in place. ECLIPSE 100 only: output set to 2 shows cumulative flows to wells and other regions and material balance errors.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>3. in addition, a balance sheet is produced for all sets of fluid-in-place regions defined with the FIP keyword.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FIPFOAM</td>
<td>Output of foam fluid in place reports (for the Foam Model):</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>1. gives a report for the whole field</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2. in addition gives a report for each fluid in place region</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FIPRESV</td>
<td>Output of region reservoir volume reports.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FIPSALT</td>
<td>Output of salt fluid in place reports (for the Brine option and salt-sensitive Polymer Flood option):</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>1. gives a report for the whole field</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2. in addition gives a report for each fluid-in-place region</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FIPSOL</td>
<td>Output of the Solvent fluid in place reports:</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>1. gives a report for the whole field</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2. gives a report for each fluid-in-place region</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FIPSURF</td>
<td>Output of surfactant fluid in place reports (for the Surfactant Model):</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>1. gives a report for the whole field</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2. in addition gives a report for each fluid-in-place region</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FIPTEMP or FIPHEAT</td>
<td>Output of energy in place reports (for the Temperature option):</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>1. gives a report for the whole field</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2. in addition gives a report for each fluid-in-place region</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FIPTR</td>
<td>Output of tracer fluid-in-place reports (for Tracer Tracking):</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>1. gives a report for the whole field</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2. in addition gives a report for each fluid-in-place region</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>-----------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>FIPVE</td>
<td>Output of fluid in place in original oil, water and gas zones (for the VE option).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>FLOGn</td>
<td>Output of inter-block component flows (for component n, where n is an integer).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FLOE</td>
<td>Output of inter-block energy flow (ECLIPSE 300 Thermal option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FLOGAS</td>
<td>Output of inter-block gas flows:</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>NNC flows are always given in ECLIPSE 300</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>=3 gives NNC flows in ECLIPSE 100</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FLOOIL</td>
<td>Output of inter-block oil flows:</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>NNC flows are always given in ECLIPSE 300</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>=3 gives NNC flows in ECLIPSE 100</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FLOSOL</td>
<td>Output of inter-block solvent flows.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FLOWAT</td>
<td>Output of inter-block water flows:</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>NNC flows are always given in ECLIPSE 300</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>=3 gives NNC flows in ECLIPSE 100</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FMISC</td>
<td>Output of relative permeability miscibility factor (Miscible option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FOAM</td>
<td>Output of grid block foam surfactant concentration (for the Foam option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FOAMADS</td>
<td>Output of grid block foam adsorption (for the Foam option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FOAMCNM</td>
<td>Output of logarithm base ten of grid block capillary number (for the Foam option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FOAMDCY</td>
<td>Output of grid block foam decay (for the Foam option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FOAMMOB</td>
<td>Output of gas mobility factor due to foam (for the Foam option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FOAMST</td>
<td>Output of oil-gas surface tension for foam.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FPC</td>
<td>Output of capillary pressure miscibility factor (Miscible option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FUGG</td>
<td>Output of component fugacities in gas (Activity Driven Diffusion).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FUGO</td>
<td>Output of component fugacities in oil (Activity Driven Diffusion).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>GASCONC</td>
<td>Output of grid-block adsorbed gas concentration (for the Coal Bed Methane option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>GASPO T</td>
<td>Output of gas potentials</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>GASSATC</td>
<td>Output of matrix grid block initial gas saturated content (for the Coal Bed Methane option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>GDOWAT</td>
<td>Output of gas displaced into oil and water zones: (for the VE option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>1= by regions and field</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>&gt;1 = by grid blocks</td>
<td></td>
<td></td>
</tr>
<tr>
<td>GENPLSTN</td>
<td>Geomechanics: outputs generalized plastic strains.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>GENPSTNS</td>
<td>Geomechanics: outputs plastic strain multiplied by stiffness matrix.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>--------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>GI</td>
<td>Output of current GI value for GI Pseudo-compositional Model. Setting the value greater than 1 will also output the cell RSGI, RVGI, BOGI and BGGI multipliers.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>GMEG</td>
<td>Output gas phase energy mobility (ECLIPSE 300 Thermal).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>GMEO</td>
<td>Output oil phase energy mobility (ECLIPSE 300 Thermal).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>GMEW</td>
<td>Output water phase energy mobility (ECLIPSE 300 Thermal).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>GOC</td>
<td>Output of current gas-oil contact (GOC) depths (for VE option): output is AAAAA if contact depth is above top of cell output is BBBBB if contact depth is below bottom of cell</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>GOCDIFF</td>
<td>Output of current G.=OC depth - initial GOC depth (for VE option).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>GRAD</td>
<td>Summary of gradient calculations for the Gradient option.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>HGAS</td>
<td>Output gas enthalpies (ECLIPSE 300 Thermal).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>HOIL</td>
<td>Output oil enthalpies (ECLIPSE 300 Thermal).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>HSOL</td>
<td>Output solid enthalpies (ECLIPSE 300 Thermal and SOLID option).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>HWAT</td>
<td>Output water enthalpies (ECLIPSE 300 Thermal).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>JV</td>
<td>Output of ln(K-Value) values.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>KRG</td>
<td>Output of grid block gas relative permeabilities.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>KRN</td>
<td>Output of grid block solvent relative permeabilities (for the Solvent Model).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>KRO</td>
<td>Output of grid block oil relative permeabilities.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>KRW</td>
<td>Output of grid block water relative permeabilities.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MLSC</td>
<td>Output of total molar density for hydrocarbon components (moles per reservoir volume).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MWAT</td>
<td>Output of total molar density for water components (moles per reservoir volume).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MULT</td>
<td>Output of grid block pore volumes and transmissibilities if they have been modified.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>NCNG</td>
<td>Output of capillary numbers for gas (VELDEP option). The normalized capillary number is output when used with keyword VDKRG.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>NCNO</td>
<td>Output of capillary numbers for oil (VELDEP option). The normalized capillary number is output when used with keyword VDKRO.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>NEWTON</td>
<td>Output of a summary of the convergence of the Newton iteration: 1. report is sent to print file. 2. in addition, a copy of the report is sent to the short form output (terminal) 3. report is sent to short form output only.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>NPMREB</td>
<td>Output of the number of passes through the rebound region during the calculations of the pore volumes of the Palmer-Mansoori rock model (keyword ROCKPAMA or ROCKPAME). Also gives the sign of the derivatives during the last calculation (positive &gt;=0). Requires CBMOPTS item 4 to be set to PMREB.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>ODGAS</td>
<td>Output of oil displaced into gas zone (for VE option)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1 = by regions and field,</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>&gt;1 = by grid blocks</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ODWAT</td>
<td>Output of oil displaced into water zone (for VE option):</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1 = by regions and field,</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>&gt;1 = by grid blocks</td>
<td></td>
<td></td>
</tr>
<tr>
<td>OILAPI</td>
<td>Output of grid block oil API values (for API Tracking).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>OILPOT</td>
<td>Output of oil potentials.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PART</td>
<td>Output of domain partitions (Parallel option).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PB or PBUB</td>
<td>Output of grid block bubble point pressures:</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RSSAT</td>
<td>1. outputs bubble point pressures only</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2. in addition, outputs the saturated dissolved gas-oil ratios</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PBLK or POLYMER</td>
<td>Output of grid block polymer concentration values (for Polymer Flood option).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PCOG</td>
<td>Output of gas-oil capillary pressures.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PCOW</td>
<td>Output of oil-water capillary pressures.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PCGW</td>
<td>Output of gas-water capillary pressures.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PD or PDEW</td>
<td>Output of grid block dew point pressures:</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RVSAT</td>
<td>1. outputs dew point pressures only</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2. in addition, outputs the saturated vaporized oil-gas ratios</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PERMSTAB</td>
<td>Geomechanics: outputs permeability multipliers (stress dependence).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PERM_MDX PERM_MDY PERM_MIZ</td>
<td>Output of directional permeability modifiers (Rock Compaction option) when RKTRMDIR or ROCKTRMX, ROCKTRMY and ROCKTRMZ are defined in ECLIPSE 300. This information can be output in ECLIPSE 100 using the ROCKC mnemonic (described below).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PERM_MOD</td>
<td>Output of permeability modifiers for Rock Compaction option (ROCKCOMP or ROCKPAME). This information can be output in E100 via the ROCKC mnemonic (described below).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PGAS</td>
<td>Output of grid block gas phase pressures</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>---------------</td>
<td>-----------------------------------------------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>PGASD or POTG</td>
<td>Output of grid block gas phase pressures (corrected to datum depth). (See also the DATUM keyword).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PKRG</td>
<td>Output of the processed scaled gas relative permeabilities at maximum gas saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the ENKRVC or ENKRVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PKRGR</td>
<td>Output of the processed scaled gas relative permeability at residual oil saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the ENKRVC or ENKRVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PKRO</td>
<td>Output of the processed scaled oil relative permeability at maximum oil saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the ENKRVC or ENKRVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PKRORG</td>
<td>Output of the processed scaled oil relative permeability at residual gas saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the ENKRVC or ENKRVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PKRORW</td>
<td>Output of the processed scaled oil relative permeability at residual water saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the ENKRVC or ENKRVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PKRW</td>
<td>Output of the processed scaled water relative permeability at maximum water saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the ENKRVC or ENKRVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PKRWR</td>
<td>Output of the processed scaled water relative permeability at residual oil saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the ENKRVC or ENKRVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PLASDISP</td>
<td>Geomechanics: outputs plastic displacements.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PLYADS</td>
<td>Polymer Flood: output of current adsorbed polymer concentration on the formation rock.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>POILD or POTO</td>
<td>Output of grid block oil phase pressures (corrected to datum depth).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>POLY</td>
<td>Output of polymer concentration (Polymer option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>POLYVM</td>
<td>Output of polymer viscosity multiplier (Polymer option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PORV_MOD</td>
<td>Output of pore volume modifiers for rock compaction (ROCKCOMP case or ROCKPAME).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>This information can be output in E100 via the ROCKC mnemonic (described below).</td>
<td></td>
<td></td>
</tr>
<tr>
<td>POTC</td>
<td>Output of initial contact corrected potential.</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

Keywords

1935
<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>PPCG</td>
<td>Output of the processed maximum gas-oil capillary pressures if</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>composition or temperature-dependent end-point scaling is selected via</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>the use of the <code>ENPCVC</code> or <code>ENPCVT</code> keywords respectively.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PPCW</td>
<td>Output of the processed maximum oil-water capillary pressures if</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>composition or temperature-dependent end-point scaling is selected via</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>the use of the <code>ENPCVC</code> or <code>ENPCVT</code> keywords respectively.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PRES or POIL</td>
<td>Output of grid block pressures.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>or PRESSURE</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PRESMIN</td>
<td>Output of minimum pressures (Irreversible rock compaction option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PRSTRESS</td>
<td>Geomechanics: outputs principal stresses: PRSTRES1, PRSTRES2,</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PRSTRES3 followed by their respective directions.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PSAT</td>
<td>Output of saturation pressures (Compositional mode).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>Note: Saturation pressures (PSAT) are not normally used in ECLIPSE</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>300, and their calculation may involve significant extra CPU time. The</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>value of PSAT depends on the cell hydrocarbon state: for single phase</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>gas it will usually be the dew point pressure; for single phase oil it</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>will usually be the bubble point pressure; for two phase cells it will</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>be the cell pressure. In thermal simulations the value of PSAT, for</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>both rock filled cells and for water filled cells, is the water</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>saturation pressure at the cell temperature.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PSGCR</td>
<td>Output of the processed scaled critical gas saturation end-points for</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>each grid cell if composition or temperature-dependent end-point</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>scaling is selected via the use of the <code>ENPTVC</code> or <code>ENPTVT</code> keywords</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>respectively or the mobile fluid correction options or <code>TZONE</code> are</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>activated.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PSGL</td>
<td>Output of the processed scaled minimum gas saturation end-points for</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>each grid cell if composition or temperature-dependent end-point</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>scaling is selected via the use of the <code>ENPTVC</code> or <code>ENPTVT</code> keywords</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>respectively.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PSGU</td>
<td>Output of the processed scaled maximum gas saturation end-points for</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>each grid cell if composition or temperature-dependent end-point</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>scaling is selected via the use of the <code>ENPTVC</code> or <code>ENPTVT</code> keywords</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>respectively.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PSOGCR</td>
<td>Output of the critical oil-in-gas saturations for each grid cell if</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>composition or temperature-dependent end-point scaling is selected via</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>the use of the <code>ENPTVC</code> or <code>ENPTVT</code> keywords respectively or the mobile</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>fluid correction options or <code>TZONE</code> are activated.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PSOWCR</td>
<td>Output of the processed scaled critical oil-in-water saturations for</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>each grid cell if composition or temperature-dependent end-point</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>scaling is selected via the use of the <code>ENPTVC</code> or <code>ENPTVT</code> keywords</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>respectively or the mobile fluid correction options or <code>TZONE</code> are</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>activated.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PSWCR</td>
<td>Output of the processed scaled critical water saturation end-points for</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>each grid cell if composition or temperature-dependent end-point</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>scaling is selected via the use of the <code>ENPTVC</code> or <code>ENPTVT</code> keywords</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>respectively or the mobile fluid correction options or <code>TZONE</code> are</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>activated.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>PSWL</td>
<td>Output of the processed scaled minimum water saturation end-points for each grid cell if composition or temperature-dependent end-point scaling is selected via the use of the <strong>ENPTVC</strong> or <strong>ENPTVT</strong> keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSWU</td>
<td>Output of the processed scaled maximum water saturation end-points for each grid cell if composition or temperature-dependent end-point scaling is selected via the use of the <strong>ENPTVC</strong> or <strong>ENPTVT</strong> keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PWAT</td>
<td>Output of grid block water phase pressures.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PWATD or POTW</td>
<td>Output of grid block water phase pressures (corrected to datum depth).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>REAC</td>
<td>Chemical reaction rates (See &quot;Chemical reactions&quot; in the <strong>ECLIPSE Technical Description</strong>).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RECOV</td>
<td>Output of recovery mechanisms (See &quot;Recovery mechanisms&quot; in the <strong>ECLIPSE Technical Description</strong>).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RESTART</td>
<td>Output of Restart files:</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>If set = 1, Restart files are created at every report time, but if the files are not unified only the last one in the run is kept (designed for fail-safe restarts).</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>If set &gt; 1, Restart files are created at every report time until this switch is reset, and all are kept.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>If set &gt; 2, the inter block flows of each phase are written to the Restart files.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>If set &gt; 3, the current fluids-in-place and phase potentials in each grid cell are written to the Restart files.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>If set = 6, Restart files are created at every timestep.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>See also the keywords <strong>RESTART, RPTRST, FMTOUT, FMTIN, UNIFIN</strong> and <strong>UNIFOUT</strong>. The value of this control will be ignored if the mnemonic <strong>BASIC</strong> in keyword <strong>RPTRST</strong> has previously been set &gt; 2.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Output of the initial Restart file (at 0.0 Days) is governed by the mnemonic <strong>RESTART</strong> in keyword <strong>RPTSOL</strong>, but there must be at least one timestep.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>In <strong>ECLIPSE 300</strong> the keyword <strong>RPTRST</strong> controls output to the Restart file.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RK</td>
<td>Output of the permeability reduction factor currently being applied to water phase relative permeability (for Polymer Flood option).</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

**Keywords**

1937
### Mnemonic | Output | E100 | E300
---|---|---|---
ROCKC | Output of pore volume and permeability modifiers for the Rock Compaction option.  
In ECLIPSE 300, this is the same as individually specifying mnemonics PORV_MOD, PERM_MOD (or PERM_MDX, PERM_MDY and PERM_MDZ in the directional case) and optionally SIGM_MOD (in a dual porosity model).  
In ECLIPSE 100, the values output (where relevant) are PRES_OVB, PRES_EFF, PEFF_MIN, PORV_MOD, PERM_MOD (or PERM_MDX, PERM_MDY and PERM_MDZ in the directional case). | x | x |
ROCKDISP | Geomechanics: outputs rock displacements ROCKDISX, ROCKDISY, ROCKDISZ. These values are defined at the faces of the grids. | x |
ROMLS | Residual oil composition (SOR option), | x |
RPORV | Output of reservoir pore volumes, | x |
PORV | Output of reservoir pore volumes at reference conditions, | x |
RS | Output of grid block solution gas-oil ratios:  
(In compositional runs this is interpreted as the GOR at surface conditions of the liquid phase in the reservoir.) | x | x |
RSW | Output of the (grid block) aqueous concentration of soluble components.  
Each component is distinguished, as with mnemonic AMF, by its associated index. | x |
RV | Output of grid block vapor oil-gas ratios.  
(In compositional runs this is interpreted as the OGR at surface conditions of what was the vapor phase in the reservoir.)  
If set >1: With the discretized matrix option active, the solution values into the matrix are also reported. | x | x |
SALT | Output of grid block salt concentration values (for the Brine option and salt-sensitive Polymer Flood option). | x | x |
SFOIL | Output of grid block foamy oil saturation. | x |
SFSOL | Output of grid block foamy solid saturation. | x |
SGAS | Output of grid block gas saturations. | x | x |
SGCRH | Output critical gas saturation (Hysteresis option). | x |
SGTRH | Output the immobile gas saturation (Hysteresis option). | x |
SGTRAP | Trapped gas saturation and dynamic trapped gas saturation (for WAG Hysteresis model only). | x | x |
SIGM_MOD | Output dual porosity sigma multipliers (Rock Compaction option) | x | x |
SL | Output liquid mole fractions (ECLIPSE 300 Thermal). | x |
SMF | Output of solid component mole fractions (CO2STORE option only). | x |
SMMULT | Output of solid mobility multiplier (SOLID option). | x |
SOIL | Output of grid block oil saturations. | x | x |
<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>SOILM</td>
<td>Mobile oil saturation (SOR option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOILR</td>
<td>Residual oil saturation (SOR option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOLADS</td>
<td>Adsorbed solid concentration (Solid option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOLADW</td>
<td>Adsorbed wetting-solid concentration (Solid option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOLWET</td>
<td>Solid wettability interpolation parameter (Solid option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SPENGAS</td>
<td>Output gas internal energy (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SPENOIL</td>
<td>Output oil internal energy (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SPENWAT</td>
<td>Output water internal energy (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SSFRAC</td>
<td>Output of suspended solids fraction (SOLID option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SSOL</td>
<td>Output of the current Solvent saturation.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SSOLID</td>
<td>Output of grid block solid saturations (SOLID option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>STATE</td>
<td>Output of states present.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>STEN</td>
<td>Output of oil/gas surface tensions (Miscible option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>STRAIN</td>
<td>Geomechanics: outputs output of strain components: STRAINXX, STRAINXY, STRAINXZ, STRAINYY, STRAINYZ, STRAINZZ. Note that these do not form a tensor - see discussion on definition of strains in &quot;Elastic stress equations&quot; in the ECLIPSE Technical Description.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SUMMARY</td>
<td>Output of solution summary at end of each timestep:</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>If set to 1: report is sent to Print file.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>If set to 2: in addition, a copy of the report is sent to the short form output (terminal).</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>If set to 3: report is sent to short form output only.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>For runs using Local Grid Refinement, adding 4 to the above reports the fluid-in-place errors in global cells due to adjacent local grids.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SURF</td>
<td>Surfactant concentration - Surfactant option.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SURFADS</td>
<td>Output of current adsorbed surfactant concentration on the formation rock (for the Surfactant Model).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SURFBLK</td>
<td>Output of the surfactant concentrations, the SURFBLK capillary number (for the Surfactant Model) and water-oil surface tension (SURFST or SURFSTES).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SURFCNM</td>
<td>log (capillary number) - Surfactant option.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SURFKR</td>
<td>Relative permeability interpolation parameter - Surfactant option.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SURFCM</td>
<td>Capillary pressure multiplier - Surfactant option.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SURFST</td>
<td>Oil-water surface tension - Surfactant option.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SV</td>
<td>Output vapor mole fractions (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SW</td>
<td>Output water mole fractions (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWAT</td>
<td>Output of grid block water saturations.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>------------</td>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>T2</td>
<td>Output temperature at the two-phase to three-phase transition (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>T3</td>
<td>Output bubble point (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TBLK or TRACER</td>
<td>Output of grid block tracer concentrations (for Tracer Tracking).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TCMULT</td>
<td>Output saturation dependent thermal conductivity transmissibility multiplier (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TEMP</td>
<td>Output of the grid block temperature (for the Temperature option or ECLIPSE 300 Thermal).</td>
<td></td>
<td>x x</td>
</tr>
<tr>
<td>TET</td>
<td>Output temperatures at last time level (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TO</td>
<td>Output temperature at which the oil phase leaves the system (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TOTCOMP</td>
<td>Output the total compressibility of the fluid system (excluding the rock).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TOTSTRES</td>
<td>Geomechanics: outputs total stress tensor: TOTSTRXX, TOTSTRXY, TOTSTRXZ, TOTSTRYY, TOTSTRZZ.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TW</td>
<td>Output temperature at which the water phase leaves the system (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TRADS or TRDCY</td>
<td>Output of tracer adsorption and decay (for Environmental Tracers).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TREATCM</td>
<td>Tracer-dependent reaction rate multipliers (See &quot;Chemical Reactions&quot; in the ECLIPSE Technical Description).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TSUB</td>
<td>Output the water saturation temperature minus the grid block temperature (THERMAL option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TUNING</td>
<td>Output of simulator control parameters set by TUNING keyword.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>UGAS</td>
<td>Output gas phase specific volumes (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>UOIL</td>
<td>Output oil phase specific volumes (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>UWAT</td>
<td>Output water phase specific volumes (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>V2</td>
<td>Output vapor mole fraction at the system two phase to three phase transition (ECLIPSE 300 Thermal).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>VFP_PROD</td>
<td>Output of vertical flow performance tables. (In order to obtain this, RPTSCHED must be placed before VFP_PROD and VFP_INJ.) This also turns on network convergence messages in the Network option. If set &gt; 1: a table is also written comparing the maximum dimensions as defined in the VFP_PDIM keyword with those observed in the input tables. If set &gt; 2: only the table comparing the maximum dimensions is written.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>VGAS</td>
<td>Output of grid block gas viscosity.</td>
<td></td>
<td>x x</td>
</tr>
<tr>
<td>VMF</td>
<td>Output of vapor mole fractions (fraction of total hydrocarbon moles in gas).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>--------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>VOIL</td>
<td>Output of grid block oil viscosity.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>VWAT</td>
<td>Output of grid block water viscosity.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>WATPOT</td>
<td>Output of water potentials.</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>
| WDOGAS   | Output of water displaced into oil and gas zones:  
(for VE option)  
1 = by regions and field,  
>1 = by grid blocks | x    |      |
| WELLS    | Output of well reports with production and injection rates and totals:  
1. gives a report of well flows  
2. gives a report of well and connection flows  
3. gives a report of layer totals  
4. gives a report of layer totals and well flows  
5. gives a report of layer totals, well and connection flows. | x    |      |
| WELSPECS or WELSPECL | Output of connection, well and group keyword data from the SCHEDULE section. | x    |      |
| WOC      | Output of current water-oil contact (WOC) depths (for VE option):  
output is AAAAA if contact depth is above top of cell  
output is BBBBB if contact depth is below bottom of cell | x    |      |
| WOCDIFF  | Output of current WOC depth - initial WOC depth (for VE option). | x    |      |
| WOCGOC   | Output of current WOC depth - current GOC depth (for VE option). | x    |      |
| XFW      | Output of liquid component mass fractions (weight fraction of each component in liquid). | x    |      |
| XGAS     | Output of gas fractional saturations (Sudation option, switched on using GRAVDR or GRAVDRM keywords). | x    |      |
| XMF      | Output of liquid component mole fractions (fraction of each component in liquid). | x    |      |
| XWAT     | Output of water fractional saturations (Sudation option, switched on using GRAVDR or GRAVDRM keywords). | x    |      |
| YFW      | Output of gas component mass fractions (weight fraction of each component in vapor). | x    |      |
| YMF      | Output of gas component mole fractions (fraction of each component in vapor). | x    |      |
| ZMF      | Output of total component mole fractions (fraction of each component in liquid). | x    |      |

*Table 3.42: RPTSCHED output controls*
Note: In ECLIPSE 300, to output the distribution of a tracer, add the name of the tracer as one of the mnemonics.

The following table lists the additional mnemonics which are available when the ODD3P relative permeability and capillary pressure hysteresis model has been selected.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>IFTGO</td>
<td>Output of gas-oil interfacial tension</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>IFTOW</td>
<td>Output of oil-water interfacial tension</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>IFTGW</td>
<td>Output of gas-water interfacial tension</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGOE</td>
<td>Output of equivalent relative permeabilities to gas for gas-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGWE</td>
<td>Output of equivalent relative permeabilities to gas for gas-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROGE</td>
<td>Output of equivalent relative permeabilities to oil for oil-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROWE</td>
<td>Output of equivalent relative permeabilities to oil for oil-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRWGE</td>
<td>Output of equivalent relative permeabilities to water for water-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRWGE</td>
<td>Output of equivalent relative permeabilities to water for water-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGOH</td>
<td>Output of turning point relative permeabilities to gas for gas-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGWH</td>
<td>Output of turning point relative permeabilities to gas for gas-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROGH</td>
<td>Output of turning point relative permeabilities to oil for oil-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROWH</td>
<td>Output of turning point relative permeabilities to oil for oil-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRWGH</td>
<td>Output of turning point relative permeabilities to water for water-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KWOH</td>
<td>Output of turning point relative permeabilities to water for water-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGOR</td>
<td>Output of representative gas-oil relative permeabilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGWR</td>
<td>Output of representative gas-water relative permeabilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROGR</td>
<td>Output of representative oil-gas relative permeabilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROWR</td>
<td>Output of representative oil-water relative permeabilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRWGR</td>
<td>Output of representative water-gas relative permeabilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRWOR</td>
<td>Output of representative water-oil relative permeabilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGOT</td>
<td>Output of opposite saturation direction turning point relative permeabilities to gas for gas-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGWT</td>
<td>Output of opposite saturation direction turning point relative permeabilities to gas for gas-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROGT</td>
<td>Output of opposite saturation direction turning point relative permeabilities to oil for oil-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROWT</td>
<td>Output of opposite saturation direction turning point relative permeabilities to oil for oil-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRWGT</td>
<td>Output of opposite saturation direction turning point relative permeabilities to water for water-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>-----------</td>
<td>------------------------------------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>KRWOT</td>
<td>Output of opposite saturation direction turning point relative permeabilities to water for water-oil system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PCOWR</td>
<td>Representative oil-water capillary pressure</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PCWOR</td>
<td>Representative water-oil capillary pressure</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PCOGR</td>
<td>Representative oil-gas capillary pressure</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PCGOR</td>
<td>Representative gas-oil capillary pressure</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PCGWR</td>
<td>Representative gas-water capillary pressure</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PCWGR</td>
<td>Representative water-gas capillary pressure</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SONRM</td>
<td>Output of normalized oil saturations</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SGNRM</td>
<td>Output of normalized gas saturations</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SWNRM</td>
<td>Output of normalized water saturations</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SDRGO</td>
<td>Output of gas phase saturation direction indicators for gas-oil system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SDRGW</td>
<td>Output of gas phase saturation direction indicators for gas-water system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SDROG</td>
<td>Output of oil phase saturation direction indicators for oil-gas system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SDROW</td>
<td>Output of oil phase saturation direction indicators for oil-water system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SDRWG</td>
<td>Output of water phase saturation direction indicators for water-gas system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SDRWO</td>
<td>Output of water phase saturation direction indicators for water-oil system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SGONE</td>
<td>Output of normalized gas equivalent opposite direction saturations for gas-oil system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SGWNE</td>
<td>Output of normalized gas equivalent opposite direction saturations for gas-water system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SOGNE</td>
<td>Output of normalized oil equivalent opposite direction saturations for oil-gas system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SOWNE</td>
<td>Output of normalized oil equivalent opposite direction saturations for oil-water system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SWGNE</td>
<td>Output of normalized water equivalent opposite direction saturations for water-gas system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SWONE</td>
<td>Output of normalized water equivalent opposite direction saturations for water-oil system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SGONH</td>
<td>Output of normalized gas hysteresis saturations for gas-oil system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SGWNH</td>
<td>Output of normalized gas hysteresis saturations for gas-water system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SOGNH</td>
<td>Output of normalized oil hysteresis saturations for oil-gas system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SOWNH</td>
<td>Normalized oil hysteresis saturations for oil-water system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SWGNH</td>
<td>Output of normalized water hysteresis saturations for water-gas system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SWONH</td>
<td>Output of normalized oil hysteresis saturations for water-oil system</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SGROB</td>
<td>Output of residual gas saturation in gas-oil system used to calculate normalized saturations</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SGRWB</td>
<td>Output of residual gas saturation in gas-water system used to calculate normalized saturations</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>-----------</td>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>SORGB</td>
<td>Output of residual oil saturation in oil-gas system used to calculate normalized saturations</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SORWB</td>
<td>Output of residual oil saturation in oil-water system used to calculate normalized saturations</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWRGB</td>
<td>Output of residual water saturation in water-gas system used to calculate normalized saturations</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWROB</td>
<td>Output of residual water saturation in water-oil system used to calculate normalized saturations</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGROP</td>
<td>Output of process dependent residual gas saturation in gas-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGRWP</td>
<td>Output of process dependent residual gas saturation in gas-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SORWP</td>
<td>Output of process dependent residual oil saturation in oil-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SORGP</td>
<td>Output of process dependent residual oil saturation in oil-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWRGP</td>
<td>Output of process dependent residual water saturation in water-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWROP</td>
<td>Output of process dependent residual water saturation in water-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGONT</td>
<td>Output of normalized gas turning point saturations for gas-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGWNT</td>
<td>Output of normalized gas turning point saturations for gas-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOGNT</td>
<td>Output of normalized oil turning point saturations for oil-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOWNT</td>
<td>Output of normalized oil turning point saturations for oil-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWGNT</td>
<td>Output of normalized water turning point saturations for water-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWONT</td>
<td>Output of normalized water turning point saturations for water-oil system</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

Table 3.43: ODD3P RPTSCHED output controls
The following table lists the deprecated mnemonics.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>SGCR</td>
<td>This mnemonic has been superseded by PSGCR (TZONE, mobile fluid correction options, ENKRV and ENKRT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGL</td>
<td>This mnemonic has been superseded by PSG (ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGU</td>
<td>This mnemonic has been superseded by PSGU (ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOGCR</td>
<td>This mnemonic has been superseded by PSGSCR (TZONE, mobile fluid correction options, ENKRV and ENKRT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOWCR</td>
<td>This mnemonic has been superseded by PSSOCR (TZONE, mobile fluid correction options, ENKRV and ENKRT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWCR</td>
<td>This mnemonic has been superseded by PSWCR (TZONE, mobile fluid correction options, ENKRV and ENKRT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWL</td>
<td>This mnemonic has been superseded by PSWL (ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWU</td>
<td>This mnemonic has been superseded by PSSU (ENKRVC and ENKRVT cases).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>WMF</td>
<td>The mnemonic has been superseded by AMF.</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

Table 3.44: Deprecated RPTSCHED output controls
Grid-valued output values in certain types of cell have the decimal point replaced with a special character for ease of identification. See 3.31 for details.

Examples

Example 1

ECLIPSE 100 example

```
RPTSCHED
'OIL' 'SGAS' 'RS' 'RESTART' 'FIP=2' 'WELSPECS' 'WELLS=2'
'SUMMARY=2' 'AQUCT' 'NEWTON' /
```

Example 2

ECLIPSE 300 example

```
RPTSCHED
PRES SWAT SOIL SGAS PSAT XMF YMF /
```
RPTSLN

Controls output to the SLN file

The keyword should be followed by a list of mnemonics which control the output of data to the streamline output files. This keyword will not cause streamlines to be traced by ECLIPSE; it will simply cause any streamlines generated for other purposes such as the Lagrangian Tracer Solver (LAGTRAC) to be output. The list should be terminated by a slash (/).

For additional control over the output, integer values greater than one may be assigned to selected mnemonics using the syntax:

```
mnemonic=integer
```

where there must be no spaces either side of the equals sign.

The following table lists the mnemonics available.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>BASIC</td>
<td>If this mnemonic is set to 1 then the standard properties will be output for each streamline. If it is set to zero, no streamline data will be output. The standard properties are the start and end ID of the streamline, the time of flight to the start and end of the streamline, phase saturations on the streamline and the phase or component number of the streamline.</td>
</tr>
<tr>
<td>FLOWS</td>
<td>If this mnemonic is present then the FLOWRATE attribute will be output. This is the molar flow of the carrier component or phase attributed to each streamline segment.</td>
</tr>
<tr>
<td>LINES</td>
<td>This mnemonic controls the number of lines output. If defaulted it will take the value 300. If set to a negative value then all lines traced will be output.</td>
</tr>
<tr>
<td>NOADDLIN</td>
<td>If this mnemonic is present then no lines traced to meet the minimum number of streamline segments requirement for any given cell will be output to the SLN files.</td>
</tr>
<tr>
<td>TRACERS</td>
<td>If this mnemonic is present then tracer concentration attributes will be output. These attributes will be named TRACXXXX where XXXX is the name of the tracer output. If a streamline is for a component or phase which does not carry a given tracer then that tracer will have a concentration of zero for that streamline</td>
</tr>
</tbody>
</table>

Table 3.45: RPTSLN output controls

Note: Separate SLN files for each report step will still be produced if UNIFOUT is specified.

Example

```
RPTSLN
   BASIC=1 LINES=1500 FLOWS /
```

Keywords

1946
**RPTSMRY**

**Controls on output from SUMMARY section**

The keyword should be followed by a single integer which determines whether or not ECLIPSE prints a table of the variables to be written to the Summary file during the current run.

A value of 0 switches the output off.

A value of 1 switches it on.

The data field should be terminated by a slash (/).

**Example**

<table>
<thead>
<tr>
<th>x</th>
<th>ECLIPSE 100</th>
<th>ECLIPSE 300</th>
<th>SPECIAL</th>
<th>RUNSPEC</th>
<th>GRID</th>
<th>EDIT</th>
<th>PROPS</th>
<th>REGIONS</th>
<th>SOLUTION</th>
<th>SUMMARY</th>
<th>SCHEDULE</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>RPTSMRY</td>
<td>1 /</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Controls on output from SOLUTION section

The keyword should be followed by a list of mnemonics which control the output of SOLUTION section data to the Print file. Wherever possible, the mnemonics correspond to the related keywords. The list should be terminated by a slash (/).

**Note:** Older ECLIPSE 100 data sets may not use mnemonics but integer controls. The meaning of these is documented in "RPTSOL" in the ECLIPSE Technical Description.

**Note:** In Parallel, ECLIPSE 100 will not output all requested quantities for performance reasons. Output can be enabled by setting item 8 of PARAOPTS to a value greater than zero.

In ECLIPSE 100

- Entering the mnemonic NOTHING clears all the reporting flags.
- For additional control over the output, integer values greater than one may be assigned to selected mnemonics using the syntax:
  
  mnemonic=integer
  
  where there must be no spaces either side of the equals sign.

**Note:** It is currently not possible to output SOLUTION section information for local grid cells using this keyword for an ECLIPSE 300 parallel run.

The following table lists the mnemonics available. Mnemonics available are identified by an x in the appropriate column.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>AIM</td>
<td>Output AIM status array (1 for implicit cells; 2 for explicit cells).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ALSURF</td>
<td>Output alkaline concentration when surfactant and alkaline models are activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ALSTML</td>
<td>Output alkaline surface tension multiplier when surfactant and alkaline models are activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>AMF</td>
<td>Output of aqueous mole fractions (CO2SOL, GASSOL and THERMAL, KWTABTn / CO2STORE and COMPW options).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ANIONS</td>
<td>Output of anion concentration.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>APIVD</td>
<td>Output of tables of initial API versus depth.</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>
### Mnemonic | Output | E100 | E300
--- | --- | --- | ---
AQUFET or AQUFETP or AQUCT or AQUFLUX or AQUANCON | Output of analytic (Fetkovich, Carter-Tracy, Constant Flux or Constant Head) aquifer data (entered using AQUFET, AQUFETP, AQUCT, AUCHWAT, AUCHGAS and AQUFLUX).
ECLIPSE 100: if set to 2: in addition, data for each aquifer-grid block connection will be printed (input using AQUANCON).
ECLIPSE 300: data for each aquifer-grid block connection will be printed (input using AQUANCON).
Note: AQUFET applies only for ECLIPSE 100. | x | x
AQANCONL | Data for each aquifer-grid block connection within local grid refinements will be printed. | x
BFORG | Output of gas Forchheimer B-Factor / phase potential difference (as described in Construction of flows in the ECLIPSE Technical Description). This is only output if VDFLOW or VDFLOWR (or VELDEP in ECLIPSE 300) has been used to activate a non-Darcy flow model.
Note that this is not the Forchheimer beta parameter. | x | x
BFORO | Output of oil Forchheimer B-Factor / phase potential difference (as described in Construction of flows in the ECLIPSE Technical Description). This is only output if VDFLOW, VDFLOWR or VELDEP has been used to activate a non-Darcy flow model.
Note that this is not the Forchheimer beta parameter. | x
BGAS | Output of gas reservoir molar densities. | x
BIOTCTAB | Geomechanics: outputs Biot’s constant multipliers (stress dependence). | x
BOIL | Output of oil reservoir molar densities. | x
BSOL | Output of solid reservoir molar densities (SOLID option). | x
BTFORG | Output of gas Forchheimer beta parameter. | x
BTFORO | Output of oil Forchheimer beta parameter. | x
BWAT | Output of water reservoir molar densities. | x
CATIONS | Output of divalent cation concentration. | x
CGAS | Output of grid-block adsorbed gas concentration (for the Coal Bed Methane option). | x
COILR | Residual oil compressibility (SOR option). | x
DENG | Output of grid block gas reservoir density. | x | x
DENO | Output of grid block oil reservoir density. | x | x
DENS | Output of grid block solid reservoir density (SOLID option). | x
DENW | Output of grid block water reservoir density. | x | x
EFFSTRES | Geomechanics: outputs initial effective in-situ stress. | x
ENERGY | Internal energy / bulk volume (THERMAL option). | x
EQUIL | Output of equilibration data. | x
<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>ESALTP</td>
<td>Output of effective salinity for polymer properties.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ESALPLY</td>
<td>Output of effective salinity for polymer properties.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ESALTS</td>
<td>Output of effective salinity for phase behavior and surfactant properties.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ESALSUR</td>
<td>Output of effective salinity for phase behavior and surfactant properties.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>FFORG</td>
<td>Forchheimer Non-Darcy flow factor for Gas (VELDEP option).</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>ECLIPSE 100: maximum deviation from Darcy Flow.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FFORO</td>
<td>Forchheimer Non-Darcy flow factor for Oil (VELDEP option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>ECLIPSE 100: not applicable in the current version.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FIP</td>
<td>Output of fluid in place reports:</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>If set to 1: initial fluids in place are reported for the whole field.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>If set to 2: in addition, initial fluids in place are reported for each fluid in place region defined with the FIPNUM keyword,</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>If set to 3: in addition, initial fluids in place are reported for all sets of fluid in place regions defined with the FIP keyword.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>ECLIPSE 300 only: FIP=1 or 2 produces no additional output as these reports are produced by default.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FIPFOAM</td>
<td>Output of foam fluid in place reports (for the Foam model):</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>1: gives a report for the whole field</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2: in addition, gives a report for each fluid-in-place region</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FIPPLY</td>
<td>Output of polymer/salt fluid in place reports (for Brine option and salt-sensitive Polymer Flood Model):</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>1: gives a report for the whole field</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2: in addition, gives a report for each fluid-in-place region</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FIPRESV</td>
<td>Output of region reservoir volume reports.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FIPSOL</td>
<td>Output of the solvent fluid in place reports:</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1: gives a report for the whole field</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2: in addition, gives a report for each fluid-in-place region</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FIPSURF</td>
<td>Output of surfactant fluid in place reports (for the Surfactant Model):</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1. gives a report for the whole field</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2. in addition, gives a report for each fluid-in-place region</td>
<td></td>
<td></td>
</tr>
<tr>
<td>FIPTEMP or FIPHEAT</td>
<td>Output of energy in place reports (for the Temperature option):</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1. gives a report for the whole field</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2. in addition, gives a report for each fluid-in-place region</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>--------</td>
<td>------</td>
<td>------</td>
</tr>
</tbody>
</table>
| FIPTR    | Output of tracer fluid in place reports (for Tracer Tracking):  
1. gives a report for the whole field  
2. in addition, gives a report for each fluid-in-place region | x    |      |
| FIPVE    | Output of fluid in place in original oil, water and gas zones in the VE option. | x    |      |
| FLOVE    | Output of fluid in place in original oil, water and gas zones in the VE option. | x    |      |
| FLOCN    | Output of inter-block component flows (for component n). | x    |      |
| FLOE     | Output of inter-block energy flow (THERMAL option only). | x    |      |
| FLOGAS   | Output of inter-block gas flows. | x  x |      |
| FLOOIL   | Output of inter-block oil flows. | x  x |      |
| FLOSOL   | Output of inter-block solvent flows (for the Solvent Model). | x    |      |
| FLOWAT   | Output of inter-block water flows. | x  x |      |
| FMISC    | Output of relative permeability miscibility factor (Miscible option). | x    |      |
| FOAM     | Output of grid block foam concentration (for the Foam option). | x  x |      |
| FOAMADS  | Output of grid block foam adsorption (for the Foam option). | x    |      |
| FOAMCNM  | Output of logarithm base ten of grid block capillary number (for the Foam option). | x  x |      |
| FOAMDcy  | Output of grid block foam decay (for the Foam option). | x    |      |
| FOAMMOB  | Output of gas mobility factor due to foam (for the Foam option). | x  x |      |
| FOAMST   | Output of gas-water surface tension. | x    |      |
| FPC      | Output of capillary pressure miscibility factor (Miscible option). | x    |      |
| FUGG     | Output of component fugacities in gas (Activity Driven Diffusion). | x    |      |
| FUGO     | Output of component fugacities in oil (Activity Driven Diffusion). | x    |      |
| GASCONC  | Output of grid-block adsorbed gas concentration (for the Coal Bed Methane option). | x    |      |
| GASPOT   | Output of gas potentials. | x    |      |
| GASSATC  | Output of matrix grid block initial gas saturated content (for the Coal Bed Methane option). | x    |      |
| GI       | Output of initial GI value for condensate/volatile oil model. | x    |      |
| GOC      | Output of initial gas-oil contact depths in the VE option:  
output is AAAAA for contact depth above top of cell  
output is BBBBB for contact depth below bottom of cell | x    |      |
<p>| GOSTRES  | Geomechanics: outputs initial gravity induced effective stress. | x    |      |
| HGAS     | Output gas enthalpy values (ECLIPSE 300 Thermal). | x    |      |
| HOIL     | Output oil enthalpy values (ECLIPSE 300 Thermal). | x    |      |
| HSOL     | Output solid enthalpy values (ECLIPSE 300 Thermal and SOLID option). | x    |      |
| HWAT     | Output water enthalpy values (ECLIPSE 300 Thermal). | x    |      |</p>
<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>JV</td>
<td>Output of ln(K-Value) values (compositional cases only).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>KRG</td>
<td>Output of grid block gas relative permeabilities.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>KRN</td>
<td>Output of grid block solvent relative permeabilities (for the Solvent Model)</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>KRO</td>
<td>Output of grid block oil relative permeabilities.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>KRW</td>
<td>Output of grid block water relative permeabilities.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>MLSC</td>
<td>Output of total molar density for hydrocarbon components (moles per reservoir volume).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>MWAT</td>
<td>Output of total molar density for water components (moles per reservoir volume).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>NCNG</td>
<td>Output of capillary numbers for gas (VELDEP option). The normalized capillary number is output when used with keyword VDKRG.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>NCNO</td>
<td>Output of capillary numbers for oil (VELDEP option). The normalized capillary number is output when used with keyword VDKRO.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>OILAPI</td>
<td>Output of initial grid block API values (for API tracking).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>OILPOT</td>
<td>Output of oil potentials.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PART</td>
<td>Output of all possible domain partitions for the Parallel option, that is the PSPLITX and PSPLITITY keywords. This is always enabled in parallel.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PERM_MDX</td>
<td>Output of directional permeability modifiers (Rock Compaction option) when RKTRMDIR or ROCKTRMX, ROCKTRMY and ROCKTRMZ are defined in ECLIPSE 300. This information can be output in ECLIPSE 100 via the ROCKC mnemonic (described below).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PERM_MDY</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PERM_MDZ</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PERM_MOD</td>
<td>Output of permeability modifiers for Rock Compaction option (ROCKCOMP or ROCKPAME). This information can be output in ECLIPSE 100 via the ROCKC mnemonic (described below).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PORV_MOD</td>
<td>Output of pore volume modifiers for rock compaction (ROCKCOMP case or ROCKPAME). This information can be output in ECLIPSE 100 via the ROCKC mnemonic (described below).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PB or PBUB</td>
<td>Output of grid block bubble point pressures: 1: outputs bubble point pressures only 2: in addition, outputs the saturated dissolved gas-oil ratios</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>RSSAT</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PBLK or POLYMER</td>
<td>Output of initial grid block polymer concentration values (for Polymer Flood Model).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PCGW</td>
<td>Output of gas-water capillary pressures (GASWAT cases).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PCOG</td>
<td>Output of gas-oil capillary pressures.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>PCOW</td>
<td>Output of oil-water capillary pressures.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PD or PDEW</td>
<td>Output of grid block dew point pressures:</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RVSAT</td>
<td>1: outputs dew point pressures only</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>2: in addition, outputs the saturated vaporized oil-gas ratios</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PERMSTAB</td>
<td>Geomechanics: outputs permeability multipliers (stress dependence).</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PGAS</td>
<td>Output of initial grid block gas phase pressures.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PGASD or POTG</td>
<td>Output of initial grid block gas phase pressures corrected to datum</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>depth) (See also the DATUM keyword).</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PGOSTRES</td>
<td>Geomechanics: outputs initial gravity induced principal stress.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PKRG</td>
<td>Output of processed max. oil relative permeability and the gas relative</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>permeability at residual oil if the end point scaling is activated.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PKRO</td>
<td>Output of processed max. oil relative permeability and the oil relative</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>permeability at residual gas and water if end-point scaling is activated.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PKRW</td>
<td>Output of processed max. water relative permeability and the water</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>relative permeability at residual oil if the end-point scaling is activated.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PKRGR</td>
<td>Output of the processed scaled gas relative permeabilities at maximum</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>gas saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the ENKRVC or ENKRVT keywords respectively.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PKRGR</td>
<td>Output of the processed scaled gas relative permeability at the residual</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>oil saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the ENKRVC or ENKRVT keywords respectively.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PKRRO</td>
<td>Output of the processed scaled oil relative permeability at the maximum</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>oil saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the ENKRVC or ENKRVT keywords respectively.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PKRO</td>
<td>Output of the processed scaled oil relative permeability at the residual</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>gas saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the ENKRVC or ENKRVT keywords respectively.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PKRO</td>
<td>Output of the processed scaled water relative permeability at the</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>maximum water saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the ENKRVC or ENKRVT keywords respectively.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PKRWR</td>
<td>Output of the processed scaled water relative permeability at the</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>residual oil saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the ENKRVC or ENKRVT keywords respectively.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>---------------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>PLYADS</td>
<td>Output of polymer concentration currently adsorbed by the rock formation (for Polymer Flood Model).</td>
<td></td>
<td></td>
</tr>
<tr>
<td>POIL</td>
<td>Output of initial grid block pressures.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>POILD or POTO</td>
<td>Output of initial grid block oil phase pressures (corrected to datum depth).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>POLY</td>
<td>Output of polymer concentration (Polymer option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>POLYVM</td>
<td>Output of polymer viscosity multiplier (Polymer option)</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>POTC</td>
<td>Output of initial contact corrected potential.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PPCG</td>
<td>Output of the processed maximum gas-oil capillary pressures (End-Point Scaling).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PPCW</td>
<td>Output of the processed maximum oil-water capillary pressures (End-Point Scaling).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PPCG</td>
<td>Output of the processed maximum gas-oil capillary pressures if composition or temperature-dependent end-point scaling selected via the use of the ENPCVC or ENPCVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PPCW</td>
<td>Output of the processed maximum oil-water capillary pressures if composition or temperature-dependent end-point scaling selected via the use of the ENPCVC or ENPCVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PRES or PRESSURE</td>
<td>Output of initial oil pressures.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>PRESMIN</td>
<td>Output of minimum pressures (irreversible Rock Compaction option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PRSTRESS</td>
<td>Geomechanics: outputs initial principal in-situ stress.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSAT</td>
<td>Output of saturation pressures (Compositional mode).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Note: The value of PSAT depends on the cell hydrocarbon state: for single phase gas it will usually be the dew point pressure; for single phase oil it will usually be the bubble point pressure; for two phase cells it will be the cell pressure. In thermal simulations the value of PSAT, for both rock filled cells and for water filled cells, is the water saturation pressure at the cell temperature.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PSGCR</td>
<td>Output of the processed scaled critical gas saturation end-points for each grid cell if composition or temperature-dependent end-point scaling is selected via the use of the ENPTVC or ENPTVT keywords respectively or the mobile fluid correction options or TZONE are activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSGL</td>
<td>Output of the processed scaled minimum gas saturation end-points for each grid cell if composition or temperature-dependent end-point scaling is selected via the use of the ENPTVC or ENPTVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSGU</td>
<td>Output of the processed scaled maximum gas saturation end-points for each grid cell if composition or temperature-dependent end-point scaling is selected via the use of the ENPTVT or ENPTVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>PSOGCR</td>
<td>Output of the processed scaled critical oil-in-gas saturations for each grid cell if composition or temperature-dependent end-point scaling is selected via the use of the ENPTV or ENPTVT keywords respectively or the mobile fluid correction options or TZONE are activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSOWCR</td>
<td>Output of the processed scaled critical oil-in-water saturations for each grid cell if composition or temperature-dependent end-point scaling is selected via the use of the ENPTVC or ENPTVT keywords respectively or the mobile fluid correction options or TZONE are activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSWCR</td>
<td>Output of the processed scaled critical water saturation end-points for each grid cell if composition or temperature-dependent end-point scaling is selected via the use of the ENPTVC or ENPTVT keywords respectively or the mobile fluid correction options or TZONE are activated.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSWL</td>
<td>Output of the processed minimum water saturation end-points for each grid cell if composition or temperature-dependent end-point scaling is selected via the use of the ENPTVC or ENPTVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PSWU</td>
<td>Output of the processed maximum water saturation end-points for each grid cell if composition or temperature-dependent end-point scaling is selected via the use of the ENPTVC or ENPTVT keywords respectively.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PTCSTRES</td>
<td>Geomechanics: outputs initial tectonic induced principal stress.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PTHSTRES</td>
<td>Geomechanics: outputs initial thermal induced principal stress.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PVDPH</td>
<td>Derivative of pore volume with respect to porosity.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PWAT</td>
<td>Output of initial grid block water phase pressures.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PWATD or POTW</td>
<td>Output of initial grid block water phase pressures (corrected to datum depth).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>QUIESC</td>
<td>Output phase pressure modifications.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RECOV</td>
<td>Output of recovery mechanisms.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RESTART</td>
<td>Output of Restart files:</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>If set &gt; 1: an initial Restart file is created.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>If set &gt; 2: the inter block flows of each phase are written to the Restart files.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>If set &gt; 3: the current fluids in place and phase potentials in each grid cell are written to the Restart files.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>(There must be at least one TIME, TSTEP or DATES keyword if an initial Restart file is required; but the simulation mode can be turned off by the keyword NOSIM in the RUNSPEC section)</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>In ECLIPSE 300 the keyword RPTRST controls output to the Restart file.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RK</td>
<td>Output of permeability reduction factor currently being applied to the water phase (for Polymer Flood Model).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>ROCKC</td>
<td>Output of pore volume and permeability modifiers for the Rock Compaction option.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>In ECLIPSE 300, this is the same as individually specifying mnemonics PORV_MOD, PERM_MOD (or PERM_MDX, PERM_MDY and PERM_MDZ in the directional case) and SIGM_MOD (in a dual porosity model).</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>In ECLIPSE 100, the values output (where relevant) are PRES_OVB, PRES_EFF, PEFF_MIN, PORV_MOD, PERM_MOD (or PERM_MDX, PERM_MDY and PERM_MDZ in the directional case).</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ROCKCON</td>
<td>Output of rock connections to the local grid refinements and to the global grid (THERMAL option only).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>ROCKMASS</td>
<td>Output rock mass (Polymer, Surfactant and Environmental Tracer options).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>ROMLS</td>
<td>Residual oil composition (SOR option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RPORV</td>
<td>Output of reservoir pore volumes.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>PORV</td>
<td>Output of reservoir pore volumes at reference pressure.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>RS</td>
<td>Output of initial grid block solution gas-oil ratios.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>In compositional runs this is interpreted as the GOR at surface conditions of the liquid phase in the reservoir.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RSV0 or PBVD</td>
<td>Output of tables of either initial ( R_s ) versus depth or initial ( P_b ) versus depth.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>RSW</td>
<td>Output of the initial aqueous concentration of soluble components. Note that this is after the initial flash calculation. Each component is distinguished, as with mnemonic ( \text{AMF} ), by its associated index.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>RV</td>
<td>Output of initial grid block vapor oil-gas ratios.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>In compositional runs this is interpreted as the OGR at surface conditions of what was the vapor phase in the reservoir.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RVVD</td>
<td>Output of tables of initial ( R_v ) versus depth.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SALT</td>
<td>Output of initial grid block salt concentration values (for Brine option and salt-sensitive Polymer Flood Model).</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SALTVD</td>
<td>Output of tables of initial salt concentration versus depth (entered using SALTVD keyword)</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>For the Multi-component Brine Model (ECLIPSE 100 only) SALTVD outputs tables of initial anion and divalent cation equivalent molar concentrations versus depth.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SFOIL</td>
<td>Output of initial grid block foamy oil saturation.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SFSOL</td>
<td>Output of initial grid block foamy solid saturation.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SGAS</td>
<td>Output of initial grid block gas saturations.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>-----------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>SGCO or SGCR or SGL or SGU</td>
<td>Output of the connate, critical and maximum gas saturations for each grid cell if the end-point scaling or the mobile fluid correction option is activated. The output is headed by mnemonic PSGCO or PSGCR or PSGL or PSGU.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SIGM_MOD</td>
<td>Output of dual porosity sigma multipliers (Rock Compaction option)</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SMMULT</td>
<td>Output of solid mobility multiplier (SOLID option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOCRS</td>
<td>Output of critical oil saturations following a surfactant flood (Surfactant option with end point scaling active).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOGCR</td>
<td>Output of the critical oil-in-gas saturations for each direction for each grid cell if the end point scaling or the mobile fluid correction is activated. The output is headed by mnemonic PSOGCR.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOIL</td>
<td>Output of initial grid block oil saturations.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SOILM</td>
<td>Mobile oil saturation (SOR option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOILR</td>
<td>Residual oil saturation (SOR option).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOLADS</td>
<td>Adsorbed solid concentration (Solid option).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOLADW</td>
<td>Adsorbed wetting-solid concentration (Solid option).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOLVD</td>
<td>Composition with depth.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOLWET</td>
<td>Solid wettability interpolation parameter (Solid option).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOWCR</td>
<td>Output of the critical oil-in-water saturations for each direction for each grid cell if the end point scaling or the mobile fluid correction is activated. The output is headed by mnemonic PSOWCR.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SSFRAC</td>
<td>Output of suspended solids fraction (SOLID option).</td>
<td></td>
<td></td>
</tr>
<tr>
<td>SSOL</td>
<td>Output of the initial solvent saturation.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SSOLID</td>
<td>Output of initial grid block solid saturations (SOLID option).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>STATE</td>
<td>Output states present.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>STEN</td>
<td>Output of oil/gas surface tensions (Miscible option).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SURF</td>
<td>Surfactant concentration - Surfactant option.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SURFADS</td>
<td>Output of current adsorbed surfactant concentration on the formation rock (for the Surfactant Model).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SURFBLK</td>
<td>Output of the Surfactant concentrations (for the Surfactant Model) and water-oil surface tensions.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SURFCNM</td>
<td>log (capillary number) - Surfactant option.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SURFKR</td>
<td>Relative permeability interpolation parameter - Surfactant option.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SURFCP</td>
<td>Capillary pressure multiplier - Surfactant option.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SURFST</td>
<td>Oil-water surface tension - Surfactant option.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SWAT</td>
<td>Output of grid block water saturations.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>SWATINIT</td>
<td>Output of scaled maximum water-oil capillary pressures generated from initial water distribution entered with keyword SWATINIT.</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>-----------</td>
<td>-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>SWCO or SWL</td>
<td>Output of the connate water saturation if the end point scaling or the mobile fluid correction option is activated. The output is headed by mnemonic PSWL.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWCR</td>
<td>Output of critical water saturation for each grid cell if the end point scaling or the mobile fluid correction option is activated. The output is headed by mnemonic PSWCR.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWU</td>
<td>Output of the maximum water saturation for each grid cell if the end point scaling or the mobile fluid correction option is activated. The output is headed by mnemonic PSWU.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TBLK or TRACER</td>
<td>Output of initial grid block tracer concentrations (for Tracer Tracking).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TCMULT</td>
<td>Saturation dependent thermal transmissibility multiplier $A$ (THERMAL option).</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TCSTRES</td>
<td>Geomechanics: outputs initial tectonic induced effective stress.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TEMP</td>
<td>Output of the grid block temperature (for the Temperature option in ECLIPSE 100 or the THERMAL option in ECLIPSE 300).</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>THPRES</td>
<td>Output of threshold pressures (see keywords EQLOPTS and THPRES)</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>THSTRES</td>
<td>Geomechanics: outputs initial thermal induced effective stress.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TRADS or TRDCY</td>
<td>Output of tracer adsorption and decay (for Environmental Tracers).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>TVDP</td>
<td>Output of initial tracer concentration versus depth tables for the Tracer Tracking option (input using TVDP).</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VGAS</td>
<td>Output of grid block gas viscosity.</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>VMF</td>
<td>Output of vapor mole fractions (fraction of total hydrocarbon moles in gas).</td>
<td></td>
<td></td>
</tr>
<tr>
<td>VOIL</td>
<td>Output of grid block oil viscosity.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>VWAT</td>
<td>Output of grid block water viscosity.</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>WATPOT</td>
<td>Output of water potentials</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>WOC</td>
<td>Output of initial water-oil contact depths in the VE option: output is AAAAA for contact depth above top of cell output is BBBBB for contact depth below bottom of cell</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>XFW</td>
<td>Output of liquid component mass fractions (weight fraction of each component in liquid).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>XGAS</td>
<td>Output of gas fractional saturations (Sudation option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>XMF</td>
<td>Output of liquid component mole fractions (fraction of each component in liquid).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>XWAT</td>
<td>Output of water fractional saturations (Sudation option).</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>YFW</td>
<td>Output of vapor component mass fractions (weight fraction of each component in gas).</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>
Mnemonic | Output | E100 | E300
---|---|---|---
YMF | Output of vapor component mole fractions (fraction of each component in gas). |  | x
SOLVD | Output of initial solution versus depth tables. |  | x
ZMF | Output of total component mole fractions. |  | x

Table 3.46: RPTSOl output controls

The following table lists the additional mnemonics which are available when the ODD3P relative permeability and capillary pressure hysteresis model has been selected.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>IFTGO</td>
<td>Output of gas-oil interfacial tension</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>IFTOW</td>
<td>Output of oil-water interfacial tension</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>IFTGW</td>
<td>Output of gas-water interfacial tension</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGOE</td>
<td>Output of equivalent relative permeabilities to gas for gas-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGWE</td>
<td>Output of equivalent relative permeabilities to gas for gas-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROGE</td>
<td>Output of equivalent relative permeabilities to oil for oil-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROWE</td>
<td>Output of equivalent relative permeabilities to oil for oil-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRWGE</td>
<td>Output of equivalent relative permeabilities to water for water-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRWOE</td>
<td>Output of equivalent relative permeabilities to water for water-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGH</td>
<td>Output of turning point relative permeabilities to gas for gas-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGWH</td>
<td>Output of turning point relative permeabilities to gas for gas-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROWH</td>
<td>Output of turning point relative permeabilities to oil for oil-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROWT</td>
<td>Output of turning point relative permeabilities to oil for oil-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGOR</td>
<td>Output of representative gas-oil relative permeabilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGWR</td>
<td>Output of representative gas-water relative permeabilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROGR</td>
<td>Output of representative oil-gas relative permeabilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROWR</td>
<td>Output of representative oil-water relative permeabilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRWR</td>
<td>Output of representative water-gas relative permeabilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRWO</td>
<td>Output of representative water-oil relative permeabilities</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGOT</td>
<td>Output of opposite saturation direction turning point relative permeabilities to gas for gas-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRGWT</td>
<td>Output of opposite saturation direction turning point relative permeabilities to gas for gas-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROGT</td>
<td>Output of opposite saturation direction turning point relative permeabilities to oil for oil-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KROWT</td>
<td>Output of opposite saturation direction turning point relative permeabilities to oil for oil-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>-----------</td>
<td>------------------------------------------------------------------------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>KRWGT</td>
<td>Output of opposite saturation direction turning point relative permeabilities to water for water-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>KRWOT</td>
<td>Output of opposite saturation direction turning point relative permeabilities to water for water-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PCOWR</td>
<td>Representative oil-water capillary pressure</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PCWOR</td>
<td>Representative water-oil capillary pressure</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PCGOR</td>
<td>Representative oil-gas capillary pressure</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PCGOR</td>
<td>Representative gas-oil capillary pressure</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PCGWR</td>
<td>Representative gas-water capillary pressure</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>PCWGR</td>
<td>Representative water-gas capillary pressure</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SORNRM</td>
<td>Output of normalized oil saturations</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGNRM</td>
<td>Output of normalized gas saturations</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWNRM</td>
<td>Output of normalized water saturations</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SDRGO</td>
<td>Output of gas phase saturation direction indicators for gas-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SDRGW</td>
<td>Output of gas phase saturation direction indicators for gas-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SDROG</td>
<td>Output of oil phase saturation direction indicators for oil-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SDROW</td>
<td>Output of oil phase saturation direction indicators for oil-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SDRWG</td>
<td>Output of water phase saturation direction indicators for water-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SDRWO</td>
<td>Output of water phase saturation direction indicators for water-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGONE</td>
<td>Output of normalized gas equivalent opposite direction saturations for gas-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGWNE</td>
<td>Output of normalized gas equivalent opposite direction saturations for gas-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOGNE</td>
<td>Output of normalized oil equivalent opposite direction saturations for oil-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOWNE</td>
<td>Output of normalized oil equivalent opposite direction saturations for oil-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWGNE</td>
<td>Output of normalized water equivalent opposite direction saturations for water-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWONE</td>
<td>Output of normalized water equivalent opposite direction saturations for water-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGONH</td>
<td>Output of normalized gas hysteresis saturations for gas-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGWNH</td>
<td>Output of normalized gas hysteresis saturations for gas-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SOGNH</td>
<td>Output of normalized oil hysteresis saturations for oil-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWNH</td>
<td>Normalized oil hysteresis saturations for oil-water system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWGNH</td>
<td>Output of normalized water hysteresis saturations for water-gas system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SWONH</td>
<td>Output of normalized oil hysteresis saturations for water-oil system</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>SGROB</td>
<td>Output of residual gas saturation in gas-oil system used to calculate normalized saturations</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>
### Mnemonic, Output, and E100, E300 Controls

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>SGRWB</td>
<td>Output of residual gas saturation in gas-water system used to calculate normalized saturations</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SORGB</td>
<td>Output of residual oil saturation in oil-gas system used to calculate normalized saturations</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SORWB</td>
<td>Output of residual oil saturation in oil-water system used to calculate normalized saturations</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SWRGB</td>
<td>Output of residual water saturation in water-gas system used to calculate normalized saturations</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SWROB</td>
<td>Output of residual water saturation in water-oil system used to calculate normalized saturations</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SGROP</td>
<td>Output of process dependent residual gas saturation in gas-oil system</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SGRWP</td>
<td>Output of process dependent residual gas saturation in gas-water system</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SORGP</td>
<td>Output of process dependent residual oil saturation in oil-gas system</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SORWP</td>
<td>Output of process dependent residual oil saturation in oil-water system</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SWRGP</td>
<td>Output of process dependent residual water saturation in water-gas system</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SWROP</td>
<td>Output of process dependent residual water saturation in water-oil system</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SGONT</td>
<td>Output of normalized gas turning point saturations for gas-oil system</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SGWNT</td>
<td>Output of normalized gas turning point saturations for gas-water system</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOGNT</td>
<td>Output of normalized oil turning point saturations for gas-oil system</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOWNT</td>
<td>Output of normalized oil turning point saturations for oil-water system</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SWGNT</td>
<td>Output of normalized water turning point saturations for water-gas system</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SWONT</td>
<td>Output of normalized water turning point saturations for water-oil system</td>
<td>x</td>
<td></td>
</tr>
</tbody>
</table>

### Table 3.47: ODD3P RPTSOL output controls

The following table lists the deprecated mnemonics.

<table>
<thead>
<tr>
<th>Mnemonic</th>
<th>Output</th>
<th>E100</th>
<th>E300</th>
</tr>
</thead>
<tbody>
<tr>
<td>SGR</td>
<td>This mnemonic has been superseded by PSGCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SGL</td>
<td>This mnemonic has been superseded by PSGL (ENKRVC and ENKRVT cases).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SGU</td>
<td>This mnemonic has been superseded by PSGU (ENKRVC and ENKRVT cases).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOGCR</td>
<td>This mnemonic has been superseded by PSOGCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SOWCR</td>
<td>This mnemonic has been superseded by PSOWCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SWCR</td>
<td>This mnemonic has been superseded by PSWCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SWL</td>
<td>This mnemonic has been superseded by PSWL (ENKRVC and ENKRVT cases).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>SWU</td>
<td>This mnemonic has been superseded by PSWU (ENKRVC and ENKRVT cases).</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Mnemonic</td>
<td>Output</td>
<td>E100</td>
<td>E300</td>
</tr>
<tr>
<td>----------</td>
<td>--------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>WMF</td>
<td>The mnemonic has been superseded by AMF.</td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

Table 3.48: Deprecated RPTSOL output controls

**Note:** For ECLIPSE 100 gas-water models in which end-point scaling is used, if SCALECRS is specified with item 1 set to YES and one or more of the mnemonics SGCO, SGCR, SGL, SGU, SGMAX, PSCGO, PSGCR, PSGL, PSGU or PPCG have been specified, the gas-water scaling limit will also be output. For a given grid block, this is the maximum of the end-point value specified via SGCR and the scaling limit specified using SCALELIM. If the SCALELIM keyword has not been specified, the gas-water scaling limit will correspond to the end-point value specified via SGCR. The gas-water scaling limit is identified by the heading G-W SLIM in the output file.

**ECLIPSE 100**

Grid-valued output values in certain types of cell have the decimal point replaced with a special character for ease of identification. See Table 3.31 for details.

**Examples**

**Example 1 (ECLIPSE 100)**

```
RPTSOL
  RESTART=2 SOIL SWAT SGAS /
```

**Example 2 (ECLIPSE 300)**

```
RPTSOL
  PRESSURE SWAT SOIL SGAS PSAT XMF YMF /
```
RS

Initial solution gas-oil ratios

The keyword should be followed by one real number for every grid block, specifying the initial solution gas-oil ratio. The data field should be terminated by a slash (/).

UNITs: sm³/sm³ (METRIC), Mscl/stb (FIELD), scc/scc (LAB), sm³/sm³ (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*2.893). Note that spaces must not be inserted on either side of the asterisk (*).

This keyword is an alternative to the PBUB keyword for defining the initial dissolved gas distribution in runs with enumerated initial conditions.

See also the keywords PRESSURE, PRVD, RV, PDEW, SWAT and SGAS in the SOLUTION section.

Example

With NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS:

<table>
<thead>
<tr>
<th>RS</th>
<th>48*1.29</th>
<th>192*1.42</th>
<th>/</th>
</tr>
</thead>
</table>

Keywords
**RCONST**

**Sets a constant Rs value for dead oil**

The keyword is used to declare that dead oil contains a constant and uniform concentration of dissolved gas. This provides a more efficient way of modeling black oil systems in which there is no free gas and the pressure never falls below the bubble point. The flags for gas and dissolved gas in the RUNSPEC section should not be switched on (keywords GAS and DISGAS), so that gas is not treated in the model as an active phase. The system therefore becomes a two-phase or single-phase model, depending on whether or not water is present as well as the oil. The oil should be treated as dead oil (using keyword PVDO to supply its PVT properties).

The keyword should be followed by a line containing two items of data, terminated by a slash (/):

1. The dissolved gas concentration (R_s).
   
   **UNITS:** sm^3/sm^3 (METRIC), Mcf/stb (FIELD), sccc/scce (LAB), sm^3/sm^3 (PVT-M)

2. The bubble point pressure.
   
   The run will terminate if the pressure in any grid block falls below this value.
   
   **UNITS:** barsa (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)

The keyword causes the oil density to be modified to include the dissolved gas, and causes the gas flow rates to be set equal to R_s * the oil flow rate.

See also keyword RCONSTT.

**Example**

```
RCONST
0.37 1015 /
```
RSCONSTT

Sets a constant Rs value for each dead oil PVT table

The keyword is used to declare that the dead oil associated with each individual PVT table region contains a constant and uniform concentration of dissolved gas. This provides a more efficient way of modeling black oil systems in which there is no free gas and the pressure never falls below the bubble point.

The difference between this keyword and RCONST is that RCONSTT allows the oil in different PVT table regions to have different Rs values. There must be no transmissibility between PVT table regions with different Rs values, to prevent oils with different Rs values from mixing. (If oil is allowed to flow between the regions, their Rs values will change with time, and the reservoir must be modeled as live oil system with gas as an active phase.) Similarly, no well can be completed in two or more PVT table regions having different Rs values.

The flags for gas and dissolved gas should not be switched on (keywords GAS and DISGAS in the RUNSPEC section), so that gas is not treated in the model as an active phase. The system therefore becomes a two-phase or single-phase model, depending on whether or not water is present as well as the oil. The oil should be treated as dead oil (using keyword PVDO to supply its PVT properties). PVT table regions are defined using keyword PVNUM.

The keyword should be followed by NTPVT (see keyword TABDIMS in the RUNSPEC section) lines, one for each PVT table region.

Each line contains two items of data, terminated by a slash (/):

1. The dissolved gas concentration (Rs).

   UNITS: \(\text{sm}^3/\text{sm}^3\) (METRIC), Mscf/stb (FIELD), scc/scc (LAB), \(\text{sm}^3/\text{sm}^3\) (PVT-M)

   The run will terminate if the pressure in any grid block in the PVT region falls below this value.

   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2. The bubble point pressure.

The keyword causes the oil density of each PVT region to be modified to include the dissolved gas, and causes the gas flow rate from each well to be set equal to its oil flow rate multiplied by the Rs value associated with the region in which it is completed. The group and field gas flows are obtained by summing the well gas flows.

Example

With NTPVT=2:

```
RSCONSTT
  0.37 1015 /
  0.47 1030 /  
```
Variation of saturated GOR with pressure and Gi

The data comprises NTPVT (item 2 in keyword TABDIMS) tables of saturated oil gas-oil ratio multiplier functions, each terminated by a slash (/), describing the variation of the maximum possible GOR of oil with pressure and Gi values. Each table consists of up to NPPVT sets of data records. Each set of records within a table consists of up to NPPVT records of data items to allow the specification of a two-dimensional (P, Gi) function which multiplies the saturated GOR value at (P, Gi=0) to give the true saturated GOR at (P, Gi). The saturated GOR value at (P, Gi=0) is obtained from the corresponding PVTO table.

The first record of each elementary set of records contains the following items of data:

1. The oil pressure value.
   The values should increase monotonically down the column.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB)

2. The factor by which the saturated oil GOR calculated from the PVTO keyword at a pressure corresponding to item 1 has to be multiplied to give the true saturated oil GOR at a pressure corresponding to item 1 and a Gi value given by item 1 in the corresponding GINODE table.

For record 2 and all subsequent records up to a maximum of NPPVT, there should be only 1 data item in the record. This item defines the corresponding factor by which the saturated oil GOR at (P, Gi=0) (computed from the PVTO data) has to be multiplied to give the true saturated oil GOR at a pressure value given in item 1 above and whose Gi value is specified in the corresponding record in the GINODE keyword.

See "Gi pseudo-compositional model" in the ECLIPSE Technical Description for further information.

Example

With NTPVT=1 and NPPVT=2

<table>
<thead>
<tr>
<th>RSGI</th>
<th>100.0</th>
<th>1.0</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.9</td>
<td>/</td>
</tr>
<tr>
<td>200.0</td>
<td>1.0</td>
<td>0.85</td>
</tr>
<tr>
<td></td>
<td></td>
<td>/</td>
</tr>
</tbody>
</table>

Keywords

1966
**Output extra data to RSM file**

This keyword requests one line per timestep output to the Run Summary file during the run. (It does not affect output to the RSM file due to the RUNSUM item).

The **RSM** keyword has no associated data.

This functionality is not available in ECLIPSE 100; the keyword is ignored.
RSSPEC

Request a restart index file

The keyword requests that a Restart Index file should be written at the end of the run. The file contains a list of the arrays written to the Restart file(s) at each restart time. See "File handling in ECLIPSE" in the ECLIPSE Technical Description.

The keyword is not required, as the restart index file is written by default. It can be disabled with the NORSSPEC keyword.

The keyword has no associated data.
Rs versus depth tables for equilibration

The data comprises NTEQUL (see keyword EQLDIMS) tables of dissolved gas-oil ratio versus depth, one for each equilibration region. Each table consists of 2 columns of data, and must be terminated by a slash (/).

1. Depth values.
   The values should increase monotonically down the column.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

2. The corresponding values of Rs, the dissolved gas-oil ratio.
   UNITS: sm^3/sm^3 (METRIC), Mcf/stb (FIELD), scf/scf (LAB), sm^3/sm^3 (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NDRXVD (see keyword EQLDIMS).

The entire table may be defaulted provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

This keyword is an alternative to the PBVD keyword, in which the bubble point pressure is tabulated against depth for each equilibration region. See also the keyword EQUIL.

In API tracking runs, however, the PBVD keyword cannot be used, and thus the RSVD keyword should be used.

The RSVD keyword may also be used in the PROPS section, but is active only for Black-Oil cases. For compositional runs please refer to alternative ways for initialization such as ZMFVD.

Note: It is possible to specify a negative depth value.

See "Initializing the study" in the ECLIPSE Technical Description for further information

Example

With NTEQUL=1 and NDRXVD ≥2:

<table>
<thead>
<tr>
<th>Depth</th>
<th>Rs</th>
</tr>
</thead>
<tbody>
<tr>
<td>7000</td>
<td>1.4</td>
</tr>
</tbody>
</table>
| 8000  | 1.4  | /  

Keywords

1969
RSW

Initial aqueous concentration of dissolved gases

This keyword should be used in CO2SOL and/or H2SSOL, or GASSOL cases where the initial solution is given by enumeration.

The keyword should be followed by up to NCMPHW real numbers for every grid block, specifying the initial concentration of dissolved gases (for example, CO₂ and/or H₂S) in the aqueous phase. This is expressed as the volume of dissolved gases (for example, CO₂ or H₂S) at surface conditions per volume of water at surface conditions. If H2SSOL is used the grid values correspond to the H₂S component. If both CO2SOL and H2SSOL are present all grid values for CO₂ are entered first followed by the values for H₂S. If GASSOL is used the grid values correspond to the NCMPHW components.

The data field should be terminated by a slash (/).

UNITs: sm³/sm³ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm³/sm³ (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). DIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*2.893). Note that spaces must not be inserted on either side of the asterisk (*).

See also the RSWVD keyword and "CO2/H2S solution in the aqueous phase" and "GASSOL solution in the aqueous phase" in the ECLIPSE Technical Description.

Examples

Example 1

    RSW
    147*0.135 /

Example 2

    RSW
    --co2--- --- h2s
    147*0.135  147*0.174/

Example 3

    RSW
    --co2---- --- h2a --- c3
    147*0.135  147*0.174 147*0.184 /
Aqueous concentration of dissolved gases versus depth

The data comprises a table of concentrations of dissolved gases (for example, CO$_2$ and or H$_2$S) versus depth. Each table consists of number columns of data (for example, 2 columns for CO2SOL or H2SSOL, 3 columns for CO2SOL and H2SSOL, and NCMPHW +1 columns for the GASSOL case), and must be terminated by a slash (/).

Column:

1. Depth values.
   
   Should increase monotonically down column.
   
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

2. Concentration of first dissolved gas (for example, CO$_2$ in the CO2SOL case or the concentration of H$_2$S for the H2SSOL case) in the aqueous phase.
   
   UNITS: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)

3. Concentration of second dissolved gas (for example, H$_2$S if both CO2SOL and H2SSOL are present) in the aqueous phase.
   
   UNITS: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)

4. to NCMPHW+1.
   
   Concentration of last dissolved gas, for example concentration of dissolved gas number NCMPHW specified by the GASSOL keyword.
   
   UNITS: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be greater than the maximum number of entries in a quantity versus depth table (EQLDIMS item 3).

If RSWVD is not used in equilibrated cases (EQUIL keyword present) then the initial concentration of dissolved gases (for example, for CO$_2$ and or H$_2$S) is taken to be zero.

The initial aqueous concentration will be set from this keyword, unless the concentration exceeds the saturated value in which case the saturated value will be taken. Note, however, that the aqueous concentration may not be in equilibrium with the gaseous concentration (for example, for CO$_2$ or H$_2$S) and hydrocarbon liquid. The concentration reported at initialization is the concentration after flashing the system, and hence dissolved gases (for example, CO$_2$ or H$_2$S) may be redistributed between the phases in each grid block.

See also “CO$_2$/H$_2$S solution in the aqueous phase” and “GASSOL solution in the aqueous phase” in the ECLIPSE Technical Description.

Examples

Example 1

Definition of the user defined concentrations of dissolved gas (for example, CO$_2$)
Example 2

Definition of the user defined concentrations of dissolved gases (for example, for CO₂ and H₂S)

Example 3

Definition of the user defined concentrations of dissolved gases in the GASSOL case.
**RTEMP**

**Specifies a constant reservoir temperature**

When an equation of state (EoS) is being used, the reservoir temperature must be specified. This is then used to calculate reduced temperatures according to the formula $T_r = T_{res}/T_c$, where $T_c$ is the critical temperature. In Thermal simulations this keyword can be used to set the initial temperature in the reservoir.

A variable temperature profile with depth can be specified with the **TEMPVD** keyword. If an explicit initialization is used, the **TEMPI** keyword can also be used.

*Note:* The keyword **RTEMPA** is accepted as an alias for **RTEMP**.

In ECLIPSE 100, the **RTEMP** keyword is used to specify the initial reservoir temperature in runs where the Temperature option is active (keyword **TEMP** in the **RUNSPEC** section). See also keywords **RTEMPVD**, **SPECHEAT**, **SPECROCK** and **WTEMP**, and "Temperature option" in the **ECLIPSE Technical Description**.

In ECLIPSE 100, the **RTEMP** keyword is accepted in the **PROPS** section only if the **EQUIL** keyword is defined in the **SOLUTION** section. Non-equilibrium initialization requires the **RTEMP** keyword to appear in the **SOLUTION** section.

Except in the following scenario, the keyword is followed by a single item of data, the reservoir temperature, terminated with a slash (/).

In ECLIPSE 300, if multiple EoS regions are used, the temperature value in the **SOLUTION** section must be entered for each such region. The number of reservoir EoS regions is specified by the 9th parameter of keyword **TABDIMS**. The use of multiple EoS regions is discussed in "Equations of State Regions" in the **ECLIPSE Technical Description**.

**UNITS:** °C (**METRIC**), °F (**FIELD**), °C (**LAB**), °C (**PVT-M**)

**DEFAULT:** 60 °F (ECLIPSE 100), 100 °C (ECLIPSE 300)

**Example**

```plaintext
RTEMP
200  /
```

---

**Keywords**

- ECLIPSE 100
- ECLIPSE 300
- SPECIAL
- RUNSPEC
- GRID
- EDIT
- PROPS
- REGIONS
- SOLUTION
- SUMMARY
- SCHEDULE

**ECLIPSE Reference Manual**

**1973**
**RTEMPA**

**Initial reservoir temperature**

The RTEMPA keyword is used to specify the initial reservoir temperature in runs where the Temperature option is active (keyword TEMP in the RUNSPEC section).

*Note:* The keyword RTEMP is accepted as an alias for RTEMPA.

See the description of keyword RTEMP for more details.

**UNITS:** °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

**DEFAULT:** 60 °F (ECLIPSE 100), 100 °C (ECLIPSE 300)

See also keywords RTEMPVD, SPECHEAT, SPECROCK and WTEMP, and "Temperature option" in the *ECLIPSE Technical Description*.

**Example**

```
RTEMPA
  200
```
**RTEMPVD**  
**Initial reservoir temperature versus depth tables**

The **RTEMPVD** keyword is used to specify the initial reservoir temperature as a function of depth, in runs where the Temperature option is active (see keyword **TEMP** in the **RUNSPEC** section).

The data comprises **NTEQUL** (see keyword **EQLDIMS**) tables of temperature versus depth, one for each equilibration region. Each table consists of 2 columns of data, and must be terminated by a slash (/).

1. **Depth values.**
   
   The values should increase monotonically down the column.
   
   **UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

2. **The corresponding values of reservoir temperature.**

   **UNITS:** °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than **NDRXVD** (see keyword **EQLDIMS**).

The entire table may be defaulted provided that the table is not the first. Defaulted tables are replaced with a copy of the previous table.

Note that this keyword sets up the initial temperature distribution, but as there are no heat sources/sinks at the top and bottom of the reservoir this thermal gradient will not be in equilibrium. Over a long time period the reservoir equilibrates to a constant temperature. However, as this equilibration process is usually very slow it may often be ignored in many reservoir applications.

See also keywords **RTEMPA**, **SPECHEAT**, **SPECROCK** and **WTEMP**, and the "Temperature option" in the **ECLIPSE Technical Description**.

**ECLIPSE 300**

The **RTEMPVD** keyword is an alias for the **TEMPVD** keyword.

**Example**

```
RTEMPVD
  1000  90
  2000  93
  3000 100 /
```

---

**Keywords**
RUNSUM

Requests tabulated output of SUMMARY data

This keyword requests that the data in the SUMMARY files should be tabulated in the Print file at the end of the run. This is exactly the same form of output as is produced by GRAF using the Run Summaries option.

Well and group control mode numbers (SUMMARY section keywords WMCTL, F/ GMCTP, F/ GMCTW, F/ GMCTG) are printed as character strings, unless the keyword LOTUS or EXCEL is used.

The operation of RUNSUM does not affect the data written to the summary files, which can still be used by GRAF in the usual manner.

The RUNSUM keyword has no associated data.

See also the keywords LOTUS, EXCEL, NARROW and SEPARATE.

Example

```
RUNSUM
```
**RV**

**Initial vapor oil-gas ratios**

The keyword should be followed by one real number for every grid block, specifying the initial vapor oil-gas ratio. The data field should be terminated by a slash (/).

**UNITS:** $\text{sm}^3/\text{sm}^3$ (METRIC), $\text{stb}/\text{Mscf}$ (FIELD), scc/scc (LAB), $\text{sm}^3/\text{sm}^3$ (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to $\text{NDIVIX}$), followed by the Y axis index (from 1 to $\text{NDIVIY}$), and finally the Z axis index (from 1 to $\text{NDIVIZ}$). $\text{DIVIX}$, $\text{NDIVIY}$ and $\text{NDIVIZ}$ are specified in RUNSPEC keyword **DIMENS**. Repeat counts may be used for repeated values (for example 115*0.00377). Note that spaces must not be inserted on either side of the asterisk (*).

This keyword is an alternative to the **PDEW** keyword for defining the initial vaporized oil distribution in runs with enumerated initial conditions.

See also the keywords **PRESSURE**, **PRVD**, **RS**, **PBUB**, **SWAT** and **SGAS** in the SOLUTION section.

**Example**

With $\text{NDIVIX}=8$, $\text{NDIVIY}=6$ and $\text{NDIVIZ}=5$ specified in RUNSPEC keyword **DIMENS**:

```plaintext
RV 48*0.00725 48*0.00326 144*0 /
```

**Keywords**

1977
Sets a constant Rv value for dry gas

The keyword is used to declare that dry gas contains a constant and uniform concentration of vaporized oil. This provides a more efficient way of modeling gas condensate systems in which there is no free oil and the pressure never falls below the dew point. The flags for oil and vaporized oil should not be switched on (keywords OIL and VAPOIL in the RUNSPEC section), so that oil is not treated in the model as an active phase. The system therefore becomes a two-phase or single-phase model, depending on whether or not water is present as well as the gas. The gas should be treated as dry gas (using keyword PVDG to supply its PVT properties).

The keyword should be followed by a line containing two items of data, terminated by a slash (/):

1. The vaporized oil concentration (Rv).
   
   UNITS: sm³/sm³ (METRIC), stb/Mscf (FIELD), scc/scc (LAB), sm³/sm³ (PVT-M)

2. The dew point pressure.
   
   DEFAULT: 0.0
   
   UNITS: bars (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

The run terminates if the pressure in any grid block falls below this value.

The keyword causes the gas density to be modified to include the vaporized oil, and causes the oil flow rates to be set equal to Rv * the gas flow rate.

See also keyword RVCONSTT.

Example

```
RVCONST 0.00047 330 /
```
RVCONSTT  Sets a constant Rv value for each dry gas PVT table

The keyword is used to declare that the dry gas associated with each individual PVT table region contains a constant and uniform concentration of vaporized oil. This provides a more efficient way of modeling gas condensate systems in which there is no free oil and the pressure never falls below the dew point.

The difference between this keyword and RVCONST is that RVCONSTT allows the gas in different PVT table regions to have different Rv values. There must be no transmissibility between PVT table regions with different Rv values, to prevent gases with different Rv values from mixing. (If gas is allowed to flow between the regions, their Rv values will change with time, and the reservoir must be modeled as wet gas system with oil as an active phase.) Similarly, no well can be completed in two or more PVT table regions having different Rv values.

The flags for oil and vaporized oil should not be switched on (keywords OIL and VAPOIL in the RUNSPEC section), so that oil is not treated in the model as an active phase. The system therefore becomes a two-phase or single-phase model, depending on whether or not water is present as well as the gas. The gas should be treated as dry gas (using keyword PVDG to supply its PVT properties). PVT table regions are defined using keyword PVTNUM.

The keyword should be followed by NTPVT (see item 2 of keyword TABDIMS in the RUNSPEC section) lines, one for each PVT table region.

Each line contains two items of data, terminated by a slash (/):

1. The vaporized oil concentration (R_v).
   
   UNITS: m^3/m^3 (METRIC), stb/Mscf (FIELD), scc/scc (LAB), m^3/m^3 (PVT-M)

2. The dew point pressure.
   
   DEFAULT: 0.0

   UNITS: barsa (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)

The run will terminate if the pressure in any grid block in the PVT region falls below this value.

The keyword causes the gas density of each PVT region to be modified to include the vaporized oil, and causes the oil flow rate from each well to be set equal to its gas flow rate multiplied by the R_v value associated with the region in which it is completed. The group and field oil flows are obtained by summing the well oil flows.

Example

With NTPVT=2:

```
RVCONSTT
0.00047 330 /
0.00057 350 /
```
Variation of saturated OGR with pressure and Gi

The data comprises NTPVT (item 2 in keyword TABDIMS) tables of saturated gas oil-gas ratio multiplier functions, each terminated by a slash (/), describing the variation of the maximum possible OGR of gas with pressure and Gi values. Each table consists of up to NPPVT sets of data records. Each set of records within a table consists of up to NPPVT records of data items to allow the specification of a two-dimensional (P, Gi) function which multiplies the saturated OGR value at (P, Gi=0) to give the true saturated OGR at (P, Gi). The saturated OGR value at (P, Gi=0) is obtained from the corresponding PVTG table.

The first record of each elementary set of records contains the following items of data:

1. The gas pressure value.
   
   The values should increase monotonically down the column.
   
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB)

2. The factor by which the saturated OGR calculated from the PVTG keyword at a pressure corresponding to item 1 has to be multiplied to give the true saturated OGR at a pressure corresponding to item 1 and a Gi value given by item 1 in the corresponding GINODE table.

For record 2 and all subsequent records up to a maximum of NPPVT, there should be only 1 data item in the record. This item defines the corresponding factor by which the saturated OGR at (P, Gi=0) (computed from the PVTG data) has to be multiplied to give the true saturated OGR for gas at a pressure value given in item 1 above and whose Gi value is specified in the corresponding record in the GINODE keyword.

See "Gi pseudo-compositional model" in the ECLIPSE Technical Description for further information.

Example

With NTPVT=1 and NPPVT=2

```
RVGI
  100.0  1.0
  0.75  /
  200.0  1.0
  0.72  /
```

Keywords

ECLIPSE Reference Manual

1980
RVVD

Rv versus depth tables for equilibration

The data comprises NTEQUL (see keyword EQLDIMS) tables of vaporized oil-gas ratio versus depth, one for each equilibration region.

Each table consists of 2 columns of data, and must be terminated by a slash (/).

1. Depth values.
   The values should increase monotonically down the column.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

2. The corresponding values of Rv, the vaporized oil-gas ratio.
   UNITS: sm$^3$/sm$^3$ (METRIC), stb/Mscf (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NDRXVD (see item 3 of keyword EQLDIMS).

The entire table may be defaulted provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

This keyword is an alternative to the PDVD keyword, in which the dew point pressure is tabulated against depth for each equilibration region. See also the keyword EQUIL.

The RVVD keyword may also be used in the PROPS section.

Note: It is possible to specify a negative depth value.

See "Initializing the study" in the ECLIPSE Technical Description for further information.

Example

With NTEQUL=1 and NDRXVD ≥2:

```plaintext
RVVD
  7000  0.00725
  8000  0.00725 /
```
S

This section contains an alphabetic listing of keywords starting with the letter S.
Regional reservoir salinity

This keyword is an extension of the SALINITY keyword and can be used to specify a salinity value region-by-region. The regions are the regions defined for the EoS (Equation Of State) using EOSNUM and their number is specified in item 9 of TABDIMS keyword.

The salinity is used in the modified Peng-Robinson equation of state in GASWAT cases and in the default CO₂ solubility data in CO2SOL cases. The units are molality (gm-M/kg).

UNITs: 10⁻³ kg-M/kg (METRIC), 10⁻³ lb-M/lb (FIELD), 10⁻³ gm-M/gm (LAB), 10⁻³ kg-M/kg (PVT-M)

The relationship to the alternative definition in terms of parts per million by mass is described in “GASWAT option” in the ECLIPSE Technical Description.

Example

Example with item 9 of TABDIMS equal to 2.

```
SALINITR
 0.51 /
 0.30 /
```
Reservoir salinity

This keyword can be used to define the salinity used in the modified Peng-Robinson equation of state in GASWAT cases and in the default CO₂ solubility data in CO2SOL cases.

The units are molality (gm-M/kg).

UNITs: \(10^{-3}\) kg-M/kg (METRIC), \(10^{-3}\) lb-M/lb (FIELD), \(10^{-3}\) gm-M/gm (LAB), \(10^{-3}\) kg-M/kg (PVT-M)

In CO2SOL cases, this keyword corrects the solubility table and does not have any impact on the surface water density. DENSITY or GRAVITY can be used to that effect.

The relationship to the alternative definition in terms of parts per million by mass is described in "GASWAT option" in the ECLIPSE Technical Description.

Example

```
SALINITY
0.51
```

Keywords

ECLIPSE Reference Manual

1984
SALT

Initial salt concentrations

The keyword should be followed by one real number for every grid block specifying the initial salt concentration. The data field should be terminated by a slash (/).

The brine option must be activated (keyword **BRINE**).

**UNITS**: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm$^3$ (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword **DIMENS**. Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*).

This keyword should be used when the initial state has been set by enumeration (see keywords **PRESSURE**, **RS**, **RV**, **SGAS** and **SWAT** in the **SOLUTION** section). For a run initialized by equilibration (**EQUIL** keyword), the **SALTVD** keyword should be used instead of **SALT**.

**ECLIPSE 100 only**

When the multi-component brine model (keyword **BRINE** in conjunction with keyword **ECLMC**) is active the **SALT** keyword specifies the initial mass concentration of every salt in the model as specified by the **BRINE** keyword. In this case the keyword should be followed by one record of data for every salt, each record containing one real number for every grid block specifying the initial salt concentration. The whole data field should be terminated by a single slash (/).

**UNITS**: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scc (LAB)

**Examples**

**Example 1**

With NDIVIX =8, NDIVIY =6 and NDIVIZ =5 specified in RUNSPEC keyword **DIMENS**:

```
SALT
48*0
48*0.4
48*2.0
48*7.0
48*20.0 /
```

**Example 2**

**ECLIPSE 100 only**

Multi-Component Brine model with two salts specified in the **BRINE** keyword and with NDIVIX =8, NDIVIY =4 and NDIVIZ =2 specified in RUNSPEC keyword **DIMENS**:

```
SALT
32*0.50  32*0.90
32*0.02  32*0.04
/
```
Effective salinity coefficients for surfactant and polymer models

This keyword may only be used when salinity dependent surfactant and/or polymer models are active (when keywords SURFSTES with ESSNODE or PLYVISCS with ESPNODE are specified in the PROPS section).

The SALTEFF keyword determines the effective salinity calculations of the water phase.

The data comprises NTPVT records of data (see item 2 of keyword TABDIMS). In a run with \( N_w \) water components (see keyword COMPW), each record requires \( 2 + 2(N_w + 1) \) items:

1. This item specifies the model for which effective salinity must be calculated. This item should be entered as characters SURF or POLY to indicate if the calculations are for Surfactant or Polymer models respectively.
2. The expression to be used for effective salinity calculation. The item should be entered as an integer number: 1 or 2.
   - If this item is set to 1, the effective salinity \( E_s \) is calculated using:
     \[
     E_s = \frac{a_0 + \sum_{i=1}^{N_w} a_i C_i}{b_0 + \sum_{i=1}^{N_w} b_i C_i}
     \]
   - If this item is set to 2, the effective salinity \( E_s \) is calculated using:
     \[
     E_s = \frac{a_0 + \sum_{i=1}^{N_w} a_i C_i}{b_0 + \frac{1}{C_s} \sum_{i=1}^{N_w} b_i C_i}
     \]
     where the coefficients \( a_0, a_i, b_0 \) and \( b_i \) are input with subsequent items of this keyword, \( C_i \) is the concentration of water component \( i \) and \( C_s \) stands for the concentration of surfactant.
     Formula 2 is only valid for the Surfactant model. If formula 2 is used for Polymer model, a warning message will be displayed and the item 2 of SALTEFF will be defaulted to 1.
3. The coefficient \( a_0 \).
4. The coefficient \( a_1 \)
   ...
   ...
   \( N_w + 3 \). The coefficient \( a_{N_w} \).
\( N_w + 4 \). The coefficient \( b_0 \)

...

...

\( 2(N_w + 1) + 2 \). The coefficient \( b_{N_w} \)

items other than the first two might be defaulted to 0.0 using 1*.

The set of coefficients input with this formula are meant to allow the user flexibility when defining the calculation of effective salinity as a function of the concentration of salts in the model. It might be noted, for instance that the expression for effective salinity input with the ECLIPSE 100 keyword \texttt{PLYESAL} can be recovered with Formula 1 above. Similarly, the expression for effective salinity input with the ECLIPSE 100 keyword \texttt{SURFESAL} can be recovered with Formula 2 above.

There are currently no restrictions on the choice of coefficients \( a_i \) and \( b_i \), \( i = 1, \ldots, N_w \). However, if item 2 is set to 2, it is not recommended to choose coefficients \( a_i \) and \( b_i \), \( i = 0, \ldots, N_w \) in such a way that the resulting expression only depends on surfactant concentration.

If the Surfactant and Polymer models are used in the same run, this keyword may be used twice; one instance of the keyword to input data for each model.

For further information on effective salinity calculations, see “Effective salinity calculations” in the \textit{ECLIPSE Technical Description} and Chapter 3 “Salinity Effect and Ion Exchange” in [Ref. 137].

**Example**

With \( N_w = 3 \) and \texttt{NTPVT}=1:

```
SALTEFF  -- model effective  a_0  a_1  a_2  a_3  b_0  b_1  b_2  b_3
         -- type salinity type 1
POLY     0.0  1.1  0.0  0.1  1.0  0.6  1.0  0.3 /

SALTEFF  -- model effective  a_0  a_1  a_2  a_3  b_0  b_1  b_2  b_3
         -- type salinity type 2
SURF     0.0  2.1  1.4  0.0  0.1  2.1  0.8  0.0 /
```
Salt concentration nodes for polymer solution viscosity

The data comprises NTPVT (see RUNSPEC keyword TABDIMS) tables of salt concentration values, each terminated by a slash (/), describing the nodal values for salt concentration to be used in the calculation of the polymer solution viscosity.

Note that this differs from the ADSALNOD keyword in being specified by PVT region rather than by saturation function.

Each table consists of up to NPPVT (see RUNSPEC keyword TABDIMS) values of the salt concentration. Within each table the values should be positive and strictly increasing.

UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scc (LAB)

The number of entries in each table must correspond to the number of polymer solution viscosity entries used in the PLYVISCS keyword.

Note that if keyword ECLMC is selected, the BRINE keyword activates the Multi-Component Brine Model (ECLIPSE 100 only) and functionality such as the polymer salt-sensitivity option (keywords POLYMER, BRINE, SALTNODE and ADSALNOD) cannot be used.

Example

With NTPVT=2 and NPPVT ≥4

```
SALTNODE
  1.0
  5.0
  10.0
  30.0 /
  5.0
  30.0 /
```
**Salt concentration versus depth for equilibration**

The data comprises NTEQUL (see keyword `EQLDIMS`) tables of salt concentration versus depth, one for each equilibration region. The Brine option must be activated (keyword `BRINE`).

Each table consists of the following columns of data, and must be terminated by a slash (/).

**Column:**

1. **Depth values.**
   - The values should increase monotonically down the column.
   - **UNITS:** m (`METRIC`), ft (`FIELD`), cm (`LAB`), m (`PVT-M`)

2. **to [N+1]** The corresponding values of salt concentration.
   - The standard Brine Model has a single salt component. In this case, column 2 of this keyword contains a single value which is the concentration of this salt at each depth.

When the Multi-Component Brine model (keyword `BRINE` with keyword `ECLMC`) is active, the `BRINE` keyword has N entries which specify the brine’s salt components. In this case, columns 2 to [N+1] of the SALTVD keyword contain tables of salt concentration versus depth, one for each salt in the model as specified by the `BRINE` keyword.

- **UNITS:** kg/sm³ (`METRIC`), lb/stb (`FIELD`), gm/scc (`LAB`), kg/sm³ (`PVT-M`)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NDRXVD (see keyword `EQLDIMS`).

See also keyword `EQUIL`.

### Examples

#### Example 1

With NTEQUL=1 and NDRXVD ≥2:

```plaintext
SALTVD
  7000  1.4
  8000  1.4 /
```

#### Example 2

**ECLIPSE 100 only**

Multi-component brine model with two salts specified in the `BRINE` keyword and with NTEQUL=2 and NDRXVD ≥2:

```plaintext
SALTVD
-- Region 1
  -- Depth Salt1 Salt2
  7000  1.4  0.02
  8000  1.4  0.03/
-- Region 2
  -- Depth Salt1 Salt2
  7000  1.2  0.01
  8000  1.3  0.02/
```

---

**Keywords**

1989
Algebraic multigrid solver

ECLIPSE 300 only. This is an optional keyword which specifies that the underlying linear systems should be solved using the algebraic multigrid solver SAMG, and controls its behavior. SAMG is currently a suitable choice in combination with the IMPES or GEOMECH keywords, but not in general for other types of simulation. Some additional SAMG control parameters are included under the CVCRIT keyword.

ECLIPSE 100 only. This keyword is only available to Schlumberger engineers and can only be accessed if the Water Services license is obtained. It is only used to replace the linear solver when using the standard solution method for a single phase tracer. It should not generally be used as it will slow down most runs. It does offer a huge benefit, though, when hydraulic dispersion, HDISP keyword, is being used. It does not work when parallel is being used.

The keyword may be followed by up to 2 items, terminated by a slash (/).

1. EPS
   This defines the tolerance used in the stopping criterion. Positive values define the required relative residual norm reduction. Negative values define the required absolute residual norm. A zero value indicates no stopping requirement.
   DEFAULT: 1.0E-4

2. REUSE
   Specifies whether to reuse the coarse grid and smoothing operators from a previous call to SAMG.
   0 do not reuse operators
   1 reuse operators throughout timestep

   DEFAULT: 1

Example

```
SAMG
1.0E-6 /
```
**SATNUM**

**Saturation function region numbers**

The keyword should be followed by one integer for every grid block in the current input box specifying the saturation function region to which it belongs. The region number should not be less than 1 or greater than NTSFUN (see keyword TABDIMS). The data must be terminated by a slash (/).

The saturation function region number specifies which set of saturation functions (input using SGFN, SOF3 and related keywords in the PROPS section) should be used to calculate relative permeabilities and capillary pressures in each grid block.

**Note:** The saturation function region numbers can be modified during the simulation by re-specifying the SATNUM keyword in the SCHEDULE section. This procedure should only be carried out with care and is not recommended in general. In particular this will not work with the parallel option.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

If the ODD3P relative permeability and capillary pressure hysteresis model has been selected, the SATNUM keyword should not be used. Instead, the primary, secondary hysteresis and tertiary hysteresis saturation region numbers should be specified using the PSTNUM, ISTNUM and DSTNUM keywords respectively.

For more information, see the saturation function keywords SGFN, SWFN, SOF2, SOF3, SOF32D, SGOF, SLGOF and SWOF in the PROPS section.

**Example**

With NTSFUN=2 specified in keyword TABDIMS; NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS; and no input BOX set:

```
SATNUM
144*1 96*2 /
```
SATOPTS

**Options for directional and hysteretic relative permeabilities**

The keyword enables the use of a directional, irreversible or hysteretic relative permeability model and an option to allow capillary pressures to depend on surface tensions. The keyword should be followed by some or all of the following options. The default is for all these options to be switched off.

The data should be terminated by a slash (/).

**DIRECT**

Directional relative permeability tables are to be used. Different saturation tables will be used for flows in the I, J and K directions. The table numbers should be entered using KRNUMX, KRNUMY and KRNUMZ in the REGIONS section. If the flow is reversible (that is the IRREVERS option is not specified), then the same table is used whether the flow is from I to I-1 or from I to I+1.

**IRREVERS**

This specifies that directional relative permeabilities are irreversible, so that different saturation tables are used depending on whether the flow is from I to I-1 or from I to I+1. The DIRECT option must also be specified here. Six sets of directional relative permeability table numbers must be defined using KRNUMX, KRNUMX-, KRNUMY, KRNUMY-, KRNUMZ and KRNUMZ- in the REGIONS section.

If DIRECT is specified but IRREVERS is not, then only 3 sets of tables are needed (KRNUMX, KRNUMY and KRNUMZ). In this case, the same table is used whether the flow is from I to I-1 or from I to I+1.

**HYSTER**

This indicates that the hysteresis option is required (see "Hysteresis" in the ECLIPSE Technical Description). Separate saturation function tables are used for drainage and imbibition. The imbibition table numbers should be identified using IMBNUM in the REGIONS section. If directional relative permeability tables are also specified (DIRECT), then imbibition tables for each of the I, J and K directions should be identified using the keywords IMBNUMX, IMBNUMY and IMBNUMZ. If in addition irreversible relative permeabilities are required (IRREVERS), then additional imbibition tables for the opposite directions should also be identified using IMBNUMX-, IMBNUMY- and IMBNUMZ-.

**SURFTENS**

This specifies that the oil-gas or oil-water capillary pressures vary with surface tension, so allowing a pressure-dependent adjustment to the oil-gas or oil-water capillary pressures. Tables of surface tension versus pressure should be entered using the STOG and STOW keywords in the PROPS section.

---

**ECLIPSE 100 only**

**ECLIPSE 300**

If the ODD3P relative permeability and capillary pressure hysteresis model has been selected, the DIRECT and IRREVERS options are not available and should not be specified. However, the SATOPTS keyword with HYSTER option must be specified in order to define the model as hysteretic.

---

**Note:** When directional forms of the keywords are used, the isotropic form is still required for use in equilibration and well block flows.

---

**Example**

Relative permeabilities are directional and irreversible:

---

**Keywords**
Keywords

1993
SAVE

Requests output of SAVE file for fast restarts

The SAVE keyword in the RUNSPEC section is used to write a SAVE file containing the initial non-recurrent data. In ECLIPSE 300, SAVE file output can be controlled using the MULTSAVE keyword.

The SAVE keyword can be used in the SCHEDULE section of either simulator to write the current solution to the SAVE file.

The SAVE file contains the data entered in the RUNSPEC, GRID, EDIT, PROPS and REGIONS sections, in a processed form so that, for example, the grid block transmissibilities do not have to be recalculated. This data is read in a fast restart. A fast restart cannot be performed without a SAVE file.

In addition for ECLIPSE 100, the time-dependent data (for example grid block solution variables and the status of wells) can optionally be written to the SAVE file instead of the Restart file, leaving the latter free for graphics output. (Use the SAVE keyword in the SCHEDULE section or the SAVE mnemonic in keyword RPTRST to control data written to the SAVE file.) The SAVE keyword must be present in the RUNSPEC section if you wish to do this, irrespective of whether you will be performing Fast Restarts or Flexible Restarts.

The SAVE keyword is required either if you subsequently want to perform a fast restart on the run, or if you wish to write the time-dependent data to the SAVE file instead of the Restart file. A SAVE file is not required for flexible restarts when the time-dependent data is written to the Restart file.

ECLIPSE 300

ECLIPSE 300 does not allow a SAVE file to be written during a parallel run. Fast restarts are not supported for parallel runs.

ECLIPSE 300

In ECLIPSE 300 the current solution can only be written to the SAVE file by using the SAVE keyword in the SCHEDULE section.

ECLIPSE 100

The SAVE keyword in the RUNSPEC section is followed by one record of data, terminated with a slash (/). The record contains a single word (in quotes):

either FORMATTED or UNFORMATTED

which sets the formatted/unformatted status of the SAVE file, irrespective of whether formatted output is specified using keyword FMTOUT in the RUNSPEC section. The default status is UNFORMATTED. The word may be shortened to F or U. The use of multiple Save files is not supported in ECLIPSE 100.

ECLIPSE 300

In ECLIPSE 300 the formatted/unformatted and multiple/unified attributes of the SAVE files are controlled by the FMTSAVE and UNIFSAVE/MULTSAVE keywords.

The SAVE keyword in the SCHEDULE section contains no items.

Examples

Example 1

Produces an unformatted SAVE file

```
  -- ECLIPSE 100
  RUNSPEC
  SAVE
  /
```
Example 2

Produces a formatted SAVE file

```plaintext
-- ECLIPSE 100
RUNSPEC
SAVE
FORMATTED /
```

Example 3

Save the current solution after 1 year

```plaintext
-- ECLIPSE 100 or ECLIPSE 300
SCHEDULE
TSTEP
365 /
SAVE
TSTEP
365 /
```
SAVEEND  Write SAVE file at end of run

This keyword requests that a SAVE file record is always written at the end of the run, irrespective of the reason for ending.

The SAVEEND keyword contains no items.
The SCALECRS keyword is optional in runs with the end-point scaling option active (keyword ENDScale in the RUNSPEC section). It is used to control the scaling method.

The keyword should be followed by one item of data, indicating whether the alternative scaling method is required, terminated with a slash (/):

either YES or NO

DEFAULT: NO

The word may be shortened to Y or N.

In the default case, the scaling process preserves relative permeabilities at two saturation nodes. The following end-points are assumed for each phase relative permeability:

\[ K_{rw}, \ SWCR \text{ and } SWU \]
\[ K_{rg}, \ SGCR \text{ and } SGU \]
\[ K_{row}, \ SOWCR \text{ and } (1.0 - SGL - SWL) \]
\[ K_{rog}, \ SOGCR \text{ and } (1.0 - SWL - SGL) \]

The alternative form of scaling (if YES is specified) preserves the relative permeabilities at three saturation nodes. In three-phase, oil-water or oil-gas runs, or runs that use the miscible flood option, the following end-points are used:

\[ K_{rw}, \ SWCR, (1.0 - SOWCR - SGL) \text{ and } SWU \]
\[ K_{rg}, \ SGCR, (1.0 - SOGCR - SWL) \text{ and } SGU \]
\[ K_{row}, \ SOWCR, (1.0 - SWCR - SGL) \text{ and } (1.0 - SWL - SGL) \]
\[ K_{rog}, \ SOGCR, (1.0 - SGCR - SWL) \text{ and } (1.0 - SWL - SGL) \]

In gas-water runs, the following end points are used if the second form of relative permeability scaling is selected:

\[ K_{rw}, \ SWCR, (1.0 - SGCR) \text{ and } SWU \]
\[ K_{rg}, \ SGCR, (1.0 - SWCR) \text{ and } SGU \]

**Note:**

If the input tables are pure normalized curves (for example if 1.0-SOWCR-SGL equals SWU in the water case), the action of SCALECRS can affect the behavior of the vertical scaling keywords (KRW and KRWR in the water case). If the default scaling is used the two end-points (1.0-SOWCR-SGL) and SWU is the same and in this case the KRWR keyword is ignored. The alternative scaling is likely to split the two end-points and hence both KRW and KRWR are honored.

If the ODD3P relative permeability and capillary pressure hysteresis model has been selected, the alternative scaling method is not available and will be ignored if specified.

See "Saturation table scaling" in the ECLIPSE Technical Description for further details.
Example

\begin{verbatim}
SCALECRS
YES /
\end{verbatim}
Sets saturation table scaling limits

The data comprises NTENDP (see item 3 of keyword ENDSCALE in the RUNSPEC section) records of saturation scaling limits to be used in the end point versus depth table method of defining saturation table end point scaling data. Each record consists of at most one real number terminated by a slash (/).

The limiting saturation scaling values must lie between 0.0 and 1.0. A default value of 0.0 is assumed for records where no scaling limit entry is specified.

SCALELIM is only applied to the water phase and can only be used in two-phase runs or runs using the Miscible Flood option.

See the keywords ENDDUM (REGIONS section) and ENPTVD (PROPS section).

See "Saturation table scaling" in the ECLIPSE Technical Description for further details.

Example

With NTENDP =6:

```
SCALELIM
0.23 /
0.16 /
0.11 /
0.10 /
/ 
/ 
```
SCDATAB

**Scale damage tables**

This keyword may be used to input tables which give the reduction factor for the PI of each connection in a well as a function of the current amount of scale deposited per unit length of perforated interval. The tables defined here are allocated to individual wells with the **WSCTAB** keyword. The rate of scale deposition around the well connections per unit water flow rate should be defined as a function of the sea water fraction in scale deposition tables supplied by the **SCDPTAB** keyword.

The keyword is followed by **NTSCDA** tables exactly, each containing two columns and no more than **NPSCDA** rows of data. The dimensions **NTSCDA** and **NPSCDA** are entered in the **RUNSPEC** section keyword **SCDPDIMS**. Each table should be terminated with a slash (/).

1. The current amount of scale deposited per unit length of perforated interval in a well connection.
   - The values should increase monotonically down the column.
   - **UNITS**: gm/m (**METRIC**), lb/ft (**FIELD**), gm/cm (**LAB**), gm/m (**PVT-M**)
2. The corresponding reduction factor for the PI of the connection.

The effect of the scale damage is to multiply the original connection transmissibility factor by this amount.

Empty tables (containing just their terminating slash) default to a copy of the previous table. Extrapolation outside the range of a table will return the value of the nearest point in the table.

---

**Note:** The scale damage tables are **not** written to the Restart files. Accordingly they must be present in any restarted data set. Thus it is also possible to change the tables at a restart.

The calculation for the effect of scale damage on a well is performed explicitly. Thus, at the end of each timestep the mass of scale deposited around each connection will be incremented according to the flow of water during the timestep and the current sea water concentration. This is translated into a PI reduction factor for each connection, which will be used during the following timestep.

The current amount of scale deposited around a connection (calculated using the scale deposition tables) is converted to the current amount of scale deposited per unit length of perforation (for use in these tables) by dividing by the connection length. For multisegment and friction wells, this length is taken directly from associated keyword data. For all other wells, this length is obtained by dividing the effective Kh value by the permeability perpendicular to the direction of penetration (see **COMPDAT** keyword).

---

**Example**

With **NTSCDA=1**, **NPSCDA** must have been set ≥ 5:

```
SCDATAB
-- scale   PI
-- deposit mult
  0.0  1.0
 10.0  0.95
 20.0  0.85
 40.0  0.55
 90.0  0.43 /
```
SCDATABG  Scale damage tables for proppant flowback

This keyword may be used to input tables which give the reduction factor for the productivity index of each connection in a well, as a function of the current amount of proppant or scale deposited per unit length of perforated interval. The tables defined here are allocated to individual wells with the WSCTAB keyword. This keyword must be used together with the PROPFLOW and GEOMECH keywords.

The keyword is followed by exactly NTSCDG tables, each containing two columns and no more than NPSCDG rows of data. The dimensions NTSCDG and NPSCDG are entered in the RUNSPEC section keyword SCDPDIMS. Each table should be terminated with a slash (/).

1. The current amount of scale deposited per unit length of perforated interval in a well connection.
   
   The values should increase monotonically down the column.

   UNITS: gm/m (METRIC), lb/ft (FIELD), gm/cm (LAB), gm/m (PVT-M)

2. The corresponding reduction factor for the PI of the connection.

The effect of the scale damage is to multiply the original connection transmissibility factor by this amount.

Empty tables (containing just their terminating slash) default to a copy of the previous table. Extrapolation outside the range of a table will return the value of the nearest point in the table.

The current amount of proppant that is deposited at each completion is calculated by accumulating the total weight of released or flowing proppant over all fracture blocks, which is then divided by the total completed length of the well. The well associated with this fracture is specified in item 12 of the PROPFLOW keyword. For multisegment and friction wells, this length is taken directly from associated keyword data. For all other wells, this length is obtained by dividing the effective Kh value by the permeability perpendicular to the direction of penetration (see COMPDAT keyword).

The calculation for the effect of scale damage on a well is performed explicitly. At the end of each timestep or report step, the mass of proppant deposited around each connection will be incremented according to the amount of proppant that is released. The resulting PI reduction factor is applied to all completion connection factors during the following timestep. When the reduction in PI is calculated for the well, the connection factors of all connections are immediately multiplied by the specified reduction factor.

Example

With NTSCDA=1, NPSCDA must have been set ≥5:

```
SCDATABG
-- scale  PI
-- deposit mult
  0.0  1.0
 10.0  0.95
 20.0  0.85
 40.0  0.55
 90.0  0.43 /
```
# SCDETAB

## Karst aquifer dissolution and transport table input to multisegment well

This keyword may be used to define tables for karst aquifer conduit pipe system, laminar and turbulent salt dissolution and transport parameters.

The `SCDETAB` keyword is followed by `NTSCDE` records, each record containing the following data items and each terminated with a slash (/). The dimension `NTSCDE` is entered in item 7 of keyword `SCDPDIMS`.

The set of records must be followed by a blank record, containing only a slash.

1. **Kinetic first order rate constant for laminar flow**
   - **UNITS:** kg/m²/day (METRIC), lb/ft²/day (FIELD), gm/cm²/hr (LAB)
   - A value of 0, or a negative value, means dissolution turned off for laminar flow
   - **DEFAULT:** 0

2. **Kinetic first order rate constant for turbulent flow**
   - **UNITS:** kg/m²/day (METRIC), lb/ft²/day (FIELD), gm/cm²/hr (LAB)
   - A value of 0, or a negative value, means dissolution turned off for turbulent flow
   - **DEFAULT:** 0

3. **Kinetic fourth order rate constant**
   - **UNITS:** kg/m²/day (METRIC), lb/ft²/day (FIELD), gm/cm²/hr (LAB)
   - A value of 0, or a negative value, means dissolution turned off for turbulent flow
   - **DEFAULT:** 0

4. **Diffusion coefficient of calcite in conduit water flow**
   - **UNITS:** m²/day (METRIC), ft²/day (FIELD), cm²/hr (LAB)
   - This input cannot be defaulted to zero, rather a positive small value should be given.
   - **DEFAULT:** 0.00001 (all unit sets)

5. **Equilibrium concentration beyond which dissolution of karst conduit will stop.**
   - **UNITS:** kg/sm³ (METRIC), lb/stb (FIELD), gm/scc (LAB)
   - **DEFAULT:** no default

6. **Relative equilibrium concentration fraction threshold beyond which dissolution kinetics changes from first order to higher order**
   - **UNITS:** dimensionless
   - **DEFAULT:** 1.0

7. **Maximum diameter size beyond which dissolution stops.**
   - **UNITS:** m (METRIC), ft (FIELD), cm (LAB)
   - **DEFAULT:** no default
8. Density of carbonate rock

UNITS: kg/m³ (METRIC), lb/ft³ (FIELD), gm/cc (LAB)

DEFAULT: default to values in the ROCKDEN keyword.

**Example**

With NTSCDE=2 specified in keyword **SCDPDIMS**:

```
SCDETAB
  1.2e-6  6.94e-9 1.38e-9 3.33e9 2.77e-9 2e-6 0.9 1*/
  1.3e-6  6.94e-9 1.38e-9 3.33e9 2.77e-9 2e-6 0.9 1*/
```
SCDPDIMS  Scale deposition and damage tables dimensions

The data consists of items that define the number of tables and the maximum number of entries in each table for the scale deposition, scale damage and scale dissolution tables. These tables are entered in the SCHEDULE section using the SCDPTAB, SCDATA and SCDATABG (ECLIPSE 300 only) keywords. (See “Scale deposition model” in the ECLIPSE Technical Description for further information.) The data must be terminated by a slash (/).

The scale damage tables for propflows are available in ECLIPSE 300 only for use with the PROPFLOW keyword.

1.  NTSCDP The number of scale deposition tables to be entered in the SCDA keyword.
    DEFAULT:  0
2.  NPSCDP The maximum number of entries in any one of the scale deposition tables.
    DEFAULT:  0
3.  NTSCDA The number of scale damage tables to be entered in the SCDATA keyword.
    DEFAULT:  0
4.  PSCDA The maximum number of entries in any one of the scale damage tables.
    DEFAULT:  0
5.  ECLIPSE 300 only for PROPFLOW. NTSCDG The number of scale damage tables to be entered in the SCDA keyword (for proppant flowback).
    DEFAULT:  0
6.  ECLIPSE 300 only for PROPFLOW NPSCDG The maximum number of entries in any one of the scale damage tables entered using SCDA keyword.
    DEFAULT:  0
7.  ECLIPSE 100 only for Karst aquifer NTSCDE The number of Karst Aquifer dissolution tables to be entered in the SCDE keyword.
    DEFAULT:  0

Example

```
SCDPDIMS
2 20 3 25 1 15 10 /
```
**SCDPTAB**

**Scale deposition tables**

This keyword may be used to input tables which give the total rate of scale deposition per unit flow rate of water into a well connection as a function of the fraction of sea water present in the water flowing through this connection. The sea water fraction is equated to the concentration of a passive water tracer which is nominated in keyword **SCDPTRAC**. The tables defined here are allocated to individual wells with the **MSCTAB** keyword. The effect of the deposited scale on the PI of the well should be defined in scale damage tables supplied by the **SCDATAB** keyword.

The keyword is followed by **NTSCDP** tables exactly, each containing two columns and no more than **NPSCDP** rows of data. The dimensions **NTSCDP** and **NPSCDP** are entered in the RUNSPEC section keyword **SCDPDIMS**. Each table should be terminated with a slash (/).

1. The fraction of sea water present in the water flowing into a well connection.
   
   The values should increase monotonically down the column.

2. The corresponding total rate of scale deposition per unit flow rate of water through the connection.
   
   **UNITS:** gm/sm$^3$ (**METRIC**), lb/stb (**FIELD**), gm/scm (**LAB**), gm/sm$^3$ (**PVT-M**)

Empty tables (containing just their terminating slash) default to a copy of the previous table. Extrapolation outside the range of a table will return the value of the nearest point in the table.

**Note:** The scale deposition tables are **not** written to the Restart files. Accordingly they must be present in any restarted data set. Thus it is also possible to change the tables at a restart.

**Example**

With **NTSCDP=2, NPSCDP ≥5**:

```plaintext
SCDPTAB
-- sea   scale
-- water deposition
-- frac rate
  0.0   0.0
  0.1   1.00
  0.15  2.00
  0.4   4.00
  0.8   9.00  /
  0.0   0.0
  0.01  0.1
  0.04  0.4
  0.12  1.00
  0.4   9.00  /
```

---

**Keywords**

2005
SCDPTTRAC  Sets a tracer to be associated with the sea water fraction for scale deposition

This keyword is used to nominate the tracer whose concentration represents the fraction of sea water present in the water flowing into a well. In conjunction with scale deposition tables (see keyword SCDPTAB), this tracer is used to calculate the current amount of scale deposited around well connections. The tracer must be already defined as a water phase passive tracer (keyword TRACER in the PROPS section), with water injectors given a tracer value of 1.0 (keyword WTRACER), while the tracer value of the water initially in the reservoir is set to 0.0. (See “Tracer tracking” in the ECLIPSE Technical Description for an account of how to define tracers.)

The keyword is followed by a single record, terminated with a slash (/).

1. Name of the tracer

   (ECLIPSE 100: Up to 3 characters, ECLIPSE 300: Up to 4 characters)

End data record with a slash (/).

**Example**

```
SCDPTTRAC
SWA /
```
Conductive fracture dimensions

The data consists of some or all of the following items, describing control parameters on the single medium conductive fractures feature active fracture, to be modeled using the single medium conductive fracture formulation (see "Dual porosity" in the ECLIPSE Technical Description for further information). The data must be terminated by a slash (/). The keyword activates the conductive fracture feature and so must be present if other conductive fracture keywords are used.

1. **MXSCFC** The maximum allowable number of fracture planar segments allowed per cell (defined using the **CONDFRAC** keyword.)
   
   DEFAULT: 5

2. **MSCBNF** The maximum number of grouping bins due to the number of fractures contained in a cell.
   
   DEFAULT: 3

3. **MSCBNF** The maximum number of grouping bins due to the value of cell permeability.
   
   DEFAULT: 3

**Example**

```
SCFDIMS
  1* 5 1 /
```
control data for secondary injection wells

the keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

the records may be terminated early with a slash (/), the remaining items taking default values. default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

the set of records must end with a blank record, containing only a slash (/).

1. secondary well name or secondary well name template.
   a template enclosed in quotes can be used to refer to multiple wells. see "well name and well list template matching" in the eclipse technical description for further details. the secondary well name must have been previously associated with a well segment of an existing multisegment well using the wsegwell keyword.

2. injector type.
   water water injector.
   stream hydrocarbon stream injector. the molar composition of the injected fluid is defined using the wellstre keyword. the name of the stream must be entered in item 9.

3. open/shut flag for the well.
   open well open for injection.
   shut well shut for injection.
   default: open.

4. control mode.
   rate controlled by surface flow rate target (item 6).

5. reserved for future use.

6. surface flow rate target or upper limit.
   units: sm³/day (metric), stb/day for oil or water (field), mscf/day for gas (field), scc/hr (lab), sm³/day (pvt-m)
   default: no target or limit.

7. reserved for future use.

8. segment pressure upper limit.
   units: barsa (metric), psia (field), atma (lab), atma (pvt-m)
   default: no upper limit.
   if the secondary well (segment) pressure rises above this value, the secondary well will be shut.

9. well stream for hydrocarbon injection.
   if injection using a hydrocarbon stream is specified in item 2, this item is required to define the name of the stream. the molar composition of the injected fluid is defined using the wellstre keyword.
Thermal

10. Temperature of injected fluid.
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
   This item is required for the Thermal option; otherwise it should be defaulted.

11. Quality of imported water.
   If the injected fluid is water, the water quality may be specified.
   UNITS: dimensionless
   DEFAULT: 1.0.

12. The maximum injection rate per unit pressure difference between the current segment pressure and the
    upper pressure limit defined by item 8.
    UNITS: sm³/day/bars (METRIC), stb/day/psi or Mscf/day/psi (FIELD), scc/hr/atm (LAB), scc/hr/atm (PVT-M)
    DEFAULT: No upper limit.

13. An alternative segment designated for maximum pressure control. In some circumstances it may be
    desirable to control the fluid injection according to a limiting maximum pressure in a segment other
    than that being used as the secondary well. If this item is specified, the injection rate will be
    constrained additionally according to the maximum pressure specified in item 14.
    DEFAULT: No maximum pressure control applied to an alternative segment.

14. The maximum pressure limit to be applied to the alternative segment number designated for maximum
    pressure control.
    UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
    DEFAULT: No upper limit.

End data record with a slash (/).
End the set of records with a blank record, containing just a slash.

Example

```
WSEGWELL
 INJ7 17  INJ7-S17 /
...
SCONINJE
 INJ7-S17 WATER OPEN RATE 1* 1000 3* 100 1 /
```
SCONPROD  Control data for secondary production wells

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated early with a slash (/). The remaining items will assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Secondary well name or secondary well name template.
   A template enclosed in quotes can be used to refer to multiple wells. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. The secondary well name must have been previously associated with a well segment of an existing multisegment well using the WSEGWELL keyword.

2. Open/shut flag for the well.
   OPEN  Well open for production.
   SHUT  Well shut for production.

   DEFAULT: OPEN.

3. Control mode.
   ORAT  Controlled by oil rate target (item 5).
   WRAT  Controlled by water rate target (item 6).
   GRAT  Controlled by gas rate target (item 7).
   LRAT  Controlled by liquid rate target (item 8).
   RESV  Controlled by reservoir fluid volume rate target (item 9).

   DEFAULT: ''

   The control mode need not be specified if the secondary well is defined to be shut.

4. Fractional rate of primary well.
   If a value is specified for this item, the fluid rate specified in item 3 will be produced at the fractional rate of the primary well, providing that the primary well is also a producer. The target specified by the control mode in item 3 is then treated as an additional limit. If this value is defaulted or if the primary well is not a producer, this item will be ignored and the well will be operated according to the target defined by item 3.

5. Oil rate target or upper limit.
   UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT–M)
   DEFAULT: No target or limit.

6. Water rate target or upper limit.
   UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT–M)
5. Gas rate target or upper limit.

In compositional mode, the gas rate of a well is defined by passing the produced well hydrocarbon through a copy of the separator to which the well is assigned, or by a flash to standard conditions if no separator has been defined.

UNITS: \( \text{sm}^3/\text{day} \) (METRIC), \( \text{Mscf/day} \) (FIELD), \( \text{scc/hr} \) (LAB), \( \text{sm}^3/\text{day} \) (PVT-M)

DEFAULT: No target or limit.

6. Liquid rate target or upper limit.

UNITS: \( \text{sm}^3/\text{day} \) (METRIC), \( \text{stb/day} \) (FIELD), \( \text{scc/hr} \) (LAB), \( \text{sm}^3/\text{day} \) (PVT-M)

DEFAULT: No target or limit.

7. Reservoir fluid volume rate target or upper limit.

The reservoir fluid volume rate is the volume of the fluids produced when measured at the average hydrocarbon pressure in the field.

UNITS: \( \text{rm}^3/\text{day} \) (METRIC), \( \text{rb/day} \) (FIELD), \( \text{rcc/hr} \) (LAB), \( \text{rm}^3/\text{day} \) (PVT-M)

DEFAULT: No target or limit.

8. Segment pressure lower limit.

UNITS: \( \text{barsa} \) (METRIC), \( \text{psia} \) (FIELD), \( \text{atma} \) (LAB), \( \text{atma} \) (PVT-M)

DEFAULT: Atmospheric pressure.

If the secondary well (segment) pressure falls below this value, the secondary well will be shut.

9. The maximum production rate per unit pressure difference between the current segment pressure and the lower pressure limit defined by item 10.

UNITS: \( \text{sm}^3/\text{day/bars} \) (METRIC), \( \text{stb/day/psi} \) or \( \text{Mscf/day/psi} \) (FIELD), \( \text{scc/hr/atm} \) (LAB), \( \text{sm}^3/\text{day/atm} \) (PVT-M)

DEFAULT: No upper limit.

10. An alternative segment designated for minimum pressure control. In some circumstances it may be desirable to control the fluid production according to a limiting minimum pressure in a segment other than that being used as the secondary well. If this item is specified, the production rate will be constrained according to the minimum pressure specified in item 13. The pressure difference between the minimum pressure limit defined by item 10 and the secondary well segment pressure is used in the flow rate calculations. In some cases, as the segment pressure approaches this limit, fluctuations in the flow rate can occur. If the alternative segment number is set equal to the secondary well segment number, item 13 can be used to define a higher cut-off pressure thereby avoiding these fluctuations.

DEFAULT: No minimum pressure control applied to an alternative segment.

11. The minimum pressure limit to be applied to the alternative segment number designated for minimum pressure control.

UNITS: \( \text{barsa} \) (METRIC), \( \text{psia} \) (FIELD), \( \text{atma} \) (LAB), \( \text{atma} \) (PVT-M)

DEFAULT: Atmospheric pressure.

End data record with a slash (/).
End the set of records with a blank record, containing just a slash.

See also keyword `WELTARG`, which can be used to reset selected control quantities individually.

**Example**

```
WSEGWELL
  PROD3 14  PROD3-S14 /
/
...
SCONPROD
  PROD3-S14 OPEN ORAT 1* 1000 4* 50 /
/
```
**Solid component compressibility**

In a run with $N_c$ components this keyword specifies the solid compressibilities at the reference pressures and temperatures. The keyword can be used in runs with solids (keyword **SOLID** in the RUNSPEC section).

The expression used to calculate component solid densities is the following:

$$
\rho_c = \frac{\rho_{ref,c}}{[1 + c_{T1,c}(T - T_{ref,c})]/[1 - c_P,c(P - P_{ref,c})]}
$$

Eq. 3.166

where

- $\rho_{ref,c}$ is the reference density, specified with keyword **SDREF**
- $c_P,c$ is the component solid compressibility, specified with this keyword
- $P_{ref,c}$ is the reference pressure, specified with keyword **SPREF**
- $T_{ref,c}$ is the reference temperature, specified with keyword **STREF**
- $c_{T1,c}$ is the thermal expansion coefficient, specified with keyword **STHERMX1**.

There should be one value for each component:

**UNITS:** 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

**DEFAULT:** 0.0

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of **TABDIMS**. The use of multiple EoS regions is discussed in "Equations of state regions" in the *ECLIPSE Technical Description*.

**Example**

For 3 components

```
SCREF
---Compressibility
5.0E-06 5.2E-6 5.4E-6 /
```
Initial coal solvent concentration versus depth tables

The data comprises NTEQUL (see keyword EQLDIMS in the RUNSPEC section) tables of initial solvent concentration versus depth, one for each equilibration region. The keyword should only be used when the Coal Bed Methane option is active (keyword COAL in RUNSPEC section). Each table consists of 2 columns of data, and must be terminated by a slash (/).

1. Depth values.
   The values should increase monotonically down the column.
   UNITS: m (METRIC), ft (FIELD), cm (LAB)

2. The corresponding values of solvent concentration in the coal.
   UNITS: sm³/m³ (METRIC), Mscf/ft³ (FIELD), scc/cc (LAB)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NDRXVD (see keyword EQLDIMS in the RUNSPEC section).

Both SOLVCONC and SCVD are entirely optional. If neither of these keywords is present, the coal solvent concentration will be taken as the equilibrium concentration for the corresponding fracture pressure.

The initial coal solvent concentration should not be set above the initial equilibrium value; ECLIPSE issues a warning if it is. If the coal solvent concentration is below the equilibrium value, the non-reversible diffusion model is recommended (set item 2 in keyword DIFFCOAL to 0.0).

See also the SOLVCONC and GCVD keywords.

Example

With NTEQUL=1 and NDRXVD ≥2

```
SCVD
  7000  94.0
  10000 94.0  / 
```
SDENSITY | Miscible gas density at surface conditions

The data comprises NTPVT (see keyword TABDIMS) records, each terminated by a slash (/). Each record consists of a single item of data, defining the surface density of the miscible injection gas.

UNITS: kg/m³ (METRIC), lb/ft³ (FIELD), gm/cc (LAB)

The SDENSITY keyword is used to specify the surface density of miscible injection gas in miscible cases (see keyword MISCIBLE) and the Solvent surface density in a Solvent case (see keyword SOLVENT) or Coal Bed Methane case (see keyword COAL). SDENSITY is an obligatory keyword in miscible flood or solvent simulations.

For the Miscible Flood Model, the gas surface density values entered under the SDENSITY keyword are used in the calculation of the effective oil and gas densities in each grid cell. The miscible gas surface density within each cell is determined by the PVT region number of the cell (see keyword PVTNUM) and not by its miscibility region number (see keyword MISCNUM).

The surface density of the solution gas is entered using either the DENSITY or GRAVITY keyword in the normal way.

**Note:** There is no facility for specifying the surface gravity (with respect to air) of the miscible gas.

**Example**

With NTPVT=3:

```
SDENSITY
  .07020 /
  .06950 /
  .06820 /
```

---

**Keywords**

2015
Reference densities for solid phase

In a run with \( N_c \) components, this keyword specifies the solid density for each component at the reference pressure and temperature. This keyword is only applicable in runs that include a solid phase (see keyword `SOLID` in the RUNSPEC section). The keyword `SDREF` should be followed by \( N_c \) real numbers, that is, a reference solid density is required for each component present in the run.

In a live-oil thermal run, the component reference solid densities are used to compute the corresponding solid densities using

\[
\rho_c = \frac{\rho_{\text{ref},c}}{\left[1 - c_p \left( P - P_{\text{ref},c} \right) \right]\left[1 + c_{T1,c} \left( T - T_{\text{ref},c} \right) \right]} \tag{3.167}
\]

where:

- \( \rho_{\text{ref},c} \) is the component reference solid density, specified with keyword `SDREF`
- \( c_p \) is the component solid compressibility, specified with keyword `SCREF`
- \( P_{\text{ref},c} \) is the component reference pressure, specified with keyword `SPREF`
- \( T_{\text{ref},c} \) is the component reference temperature, specified with keyword `STREF`
- \( c_{T1,c} \) is the component thermal expansion coefficient, specified with keyword `STHERMX1`.

Regarding the reference densities specified using `SDREF`:

- **UNITS**: kg/m\(^3\) (METRIC), lb/ft\(^3\) (FIELD), gm/cc (LAB), kg/m\(^3\) (PVT-M)
- **DEFAULT**: 799.2112 kg/m\(^3\) (METRIC) for each component, except for the `CO2STORE` options where components NaCl, CaCl\(_2\) and CaCO\(_3\) are assigned default values of 2170 kg/m\(^3\), 2150 kg/m\(^3\) and 2709.89 kg/m\(^3\), respectively (METRIC).

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 in keyword `TABDIMS`. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

Examples

**Example 1**

For a run containing 5 components:

```
SDREF
2*  43.887  2*  /
```

**Example 2**

For a Blackoil model with oil, gas and one solid component:
EDREF
2* 43.887 /
**Initial gas to oil saturation direction**

This keyword specifies the initial gas to oil saturation direction for each grid cell. This keyword is only used for the ODD3F option and is only required if the initial saturation direction is to be initialized to a value other than the default, primary saturation direction.

Each SDRGO keyword should be followed by one integer number for each grid cell in the current input box specifying the initial saturation direction for the grid cell. The values should be in the range of 1 to 4 inclusive. If the value of SDRGO is omitted for any grid cell, the default value of 1 will be assigned. The following values are permitted:

1. Primary saturation direction with transfer to hysteresis directions not permitted.
2. Primary saturation direction with transfer to hysteresis directions permitted.
4. Tertiary hysteresis saturation direction.

See "Saturation table scaling" in the *ECLIPSE Technical Description* for further details.

See also the **SDROG, SDROW, SDRWO, SDRGW** and **SDRWG** keywords.

**Example**

With NDIVIX=3, NDIVIY=4 and NDIVIZ=5 specified in RUNSPEC keyword **DIMENS** and no input BOX set:

```
SDRGO
12*1 12*2 12*3 24*4/
```
SDRGW

Initial gas to water saturation direction

This keyword specifies the initial gas to water saturation direction for each grid cell. This keyword is only used for the *ODD3P* option and is only required if the initial saturation direction is to be initialized to a value other than the default, primary saturation direction.

Each *SDRGW* keyword should be followed by one integer number for each grid cell in the current input box specifying the initial saturation direction for the grid cell. The values should be in the range of 1 to 4 inclusive. If the value of *SDRGW* is omitted for any grid cell, the default value of 1 will be assigned. The following values are permitted:

1. Primary saturation direction with transfer to hysteresis directions not permitted.
2. Primary saturation direction with transfer to hysteresis directions permitted.
4. Tertiary hysteresis saturation direction.

See "Saturation table scaling" in the *ECLIPSE Technical Description* for further details.

See also the *SDROG, SDRGO, SDROW, SDRWO* and *SDRWG* keywords.

Example

With *NDIVIX=3, NDIVIY=4* and *NDIVIZ=5* specified in RUNSPEC keyword *DIMENS* and no input BOX set:

```
  SDRGW
  12*1 12*2 12*3 24*4/
```

Keywords

2019
**SDROG**

**Initial oil to gas saturation direction**

This keyword specifies the initial oil to gas saturation direction for each grid cell. This keyword is only used for the ODD3F option and is only required if the initial saturation direction is to be initialized to a value other than the default, primary saturation direction.

Each SDROG keyword should be followed by one integer number for each grid cell in the current input box specifying the initial saturation direction for the grid cell. The values should be in the range of 1 to 4 inclusive. If the value of SDROG is omitted for any grid cell, the default value of 1 will be assigned. The following values are permitted:

1. Primary saturation direction with transfer to hysteresis directions not permitted.
2. Primary saturation direction with transfer to hysteresis directions permitted.
4. Tertiary hysteresis saturation direction.

See "Saturation table scaling" in the ECLIPSE Technical Description for further details.

See also the SDRGO, SDROW, SDRWO, SDRGW and SDRWG keywords.

**Example**

With NDIVIX=3, NDIVIY=4 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS and no input BOX set:

```
SDROG
  12*1 12*2 12*3 24*4/
```
**Initial oil to water saturation direction**

This keyword specifies the initial oil to water saturation direction for each grid cell. This keyword is only used for the **ODD3F** option and is only required if the initial saturation direction is to be initialized to a value other than the default, primary saturation direction.

Each **SDROW** keyword should be followed by one integer number for each grid cell in the current input box specifying the initial saturation direction for the grid cell. The values should be in the range of 1 to 4 inclusive. If the value of **SDROW** is omitted for any grid cell, the default value of 1 will be assigned. The following values are permitted:

1. Primary saturation direction with transfer to hysteresis directions not permitted.
2. Primary saturation direction with transfer to hysteresis directions permitted.
4. Tertiary hysteresis saturation direction.

See "Saturation table scaling" in the *ECLIPSE Technical Description* for further details.

See also the **SDROG, SDRGO, SDRWO, SDRGW** and **SDRWG** keywords.

**Example**

With **NDIVIX=3**, **NDIVIY=4** and **NDIVIZ=5** specified in RUNSPEC keyword **DIMENS** and no input **BOX** set:

```
  SDROW
  12*1 12*2 12*3 24*4/
```
### Initial water to gas saturation direction

This keyword specifies the initial water to gas saturation direction for each grid cell. This keyword is only used for the **ODD3P** option and is only required if the initial saturation direction is to be initialized to a value other than the default, primary saturation direction.

Each **SDRWG** keyword should be followed by one integer number for each grid cell in the current input box specifying the initial saturation direction for the grid cell. The values should be in the range of 1 to 4 inclusive. If the value of **SDRWG** is omitted for any grid cell, the default value of 1 will be assigned. The following values are permitted:

1. Primary saturation direction with transfer to hysteresis directions not permitted.
2. Primary saturation direction with transfer to hysteresis directions permitted.
4. Tertiary hysteresis saturation direction.

See "Saturation table scaling" in the *ECLIPSE Technical Description* for further details.

See also the **SDROG, SDRGO, SDROW, SDRWO** and **SDRGW** keywords.

### Example

With NDIVIX=3, NDIVIY=4 and NDIVIZ=5 specified in RUNSPEC keyword **DIMENS** and no input BOX set:

```
  SDRWG 12*1 12*2 12*3 24*4/
```

---

Keywords

2022
**SDRWO**

**Initial water to oil saturation direction**

This keyword specifies the initial water to oil saturation direction for each grid cell. This keyword is only used for the **ODD3P** option and is only required if the initial saturation direction is to be initialized to a value other than the default, primary saturation direction.

Each **SDRWO** keyword should be followed by one integer number for each grid cell in the current input box specifying the initial saturation direction for the grid cell. The values should be in the range of 1 to 4 inclusive. If the value of **SDRWO** is omitted for any grid cell, the default value of 1 will be assigned. The following values are permitted:

1. Primary saturation direction with transfer to hysteresis directions not permitted.
2. Primary saturation direction with transfer to hysteresis directions permitted.
4. Tertiary hysteresis saturation direction.

See "Saturation table scaling" in the **ECLIPSE Technical Description** for further details.

See also the **SDROG**, **SDRGO**, **SDROW**, **SDRGW** and **SDRWG** keywords.

**Example**

With **NDIVIX=3, NDIVIY=4** and **NDIVIZ=5** specified in **RUNSPEC** keyword **DIMENS** and no input BOX set:

```plaintext
SDRWO
12*1 12*2 12*3 24*4/
```

---

**Keywords**

---

2023


**Sector boundary conditions**

This keyword is used to specify or create the boundary conditions for a SECTOR run. A sector model can be run with either no-flow boundary conditions or with flux boundary conditions. Flux boundary conditions can be created from existing full field simulation output files.

The keyword contains the following items of data, terminated with a slash (/):

1. **The type of boundary conditions**
   - **NOFLOW**  No flow on the sector boundaries.
   - **DUMPFLUX** Generate the fluxes which are stored in a FLUX file.
   - **USEFLUX** Use fluxes from an existing FLUX file on the sector boundaries.

   **DEFAULT:** NOFLOW

2. **The name of the full field run.** This information is not required if the first item is NOFLOW.

   The no-flow boundary condition can be applied without any extra data.

   For flux boundary conditions, a flux file must first be generated from an existing full field run. This step is performed by running a sector model with the DUMPFLUX boundary condition. The flux file is generated from the full field RESTART files. (A GRID or EGRID file of the full field simulation is also required.) This intermediate step is relatively quick, since no simulation is performed.

   To perform a sector run with flux boundaries, the USEFLUX boundary condition should be used. Data from the flux file previously created by the DUMPFLUX run is read in and interpolated between report steps.

   Running a sector model with flux boundary conditions is similar to the approach using FLUXNUM regions, described in "Flux boundary conditions" in the ECLIPSE Technical Description. There are, however, two important differences:

   - A sector can be specified after a full field simulation has been run. Flux files can be generated from a full field run for any number of sectors. On the other hand, when flux boundary conditions are generated for FLUXNUM regions, the regions must be specified before the full field run is performed.
   - The sector flux file only contains data at the report steps of the full field run. This is then interpolated during the sector runs. Flux files that are created during a full field run for FLUXNUM regions contain data for each timestep.

   These differences mean the sector model is a more flexible method than using FLUXNUM regions, but slightly less accurate.

   **Note:** The sector flux boundary condition cannot be used together with FLUXNUM.

   **Note:** The sector boundary should not cross LGRs, when the FLUX boundary option is used.

### Example with NOFLOW

To perform a simulation on an 11 x 11 x 30 sector of a full field model, the following keywords can be added to the RUNSPEC section of the data set:
Example creating the FLUX file

If a full field model (BASE.DAT) has been run, a flux file can be generated for a sector of the full field. A copy of the full field data file is made, with the following keywords added to the RUNSPEC section:

```
-- Specify sector
SECTOR
  30 40 30 40 1 30 /
-- Generate flux file for sector from the full field run
SECTBC
  DUMPFLUX BASE /
```

When ECLIPSE 300 is run with this data file a simulation is not performed. Instead it uses the RESTART files and the GRID or EGRID file from the BASE run to produce a flux file:

BASE30-40X30-40Y.FLUX (or FFLUX if using formatted output).

If the output from the full field run is in a different directory, then the path should be specified with the file name, for example:

```
-- Specify sector
SECTOR
  30 40 30 40 1 30 /
-- Generate flux file for sector from the full field run
SECTBC
  DUMPFLUX '../BASE' /
```

A flux file will then be generated in the same directory as the full field output:

`../BASE30-40X30-40Y.FLUX` (or FFLUX if using formatted output).

*Note:* The sector size is part of the flux file name, so several flux files for different sectors can be present.

Example using the FLUX file

To run a sector model with flux boundary conditions a copy of the full field data file is made, with the following keywords added to the RUNSPEC section:

```
-- Specify sector
SECTOR
  30 40 30 40 1 30 /
-- Use the fluxes on file as boundary conditions for sector
SECTBC
  USEFLUX BASE/
```

This will take the fluxes from the file

BASE30-40X30-40Y.FLUX (or FFLUX if using formatted input).
**Sector model**

This keyword specifies a sector of the full field model. Simulation is only performed on the sector. This allows the user to perform quick simulations on small areas of the full field model, with smaller memory requirements and without the need to rewrite the data set.

Boundary conditions for the sector model are given by the `SECTBC` keyword.

The keyword contains the following items of data:

1. First block on X axis of the new input box (\(IX1\))
2. Last block on X axis of the new input box (\(IX2\))
3. First block on Y axis of the new input box (\(JY1\))
4. Last block on Y axis of the new input box (\(JY2\))
5. First block on Z axis of the new input box (\(KZ1\))
6. Last block on Z axis of the new input box (\(KZ2\))

The data must satisfy:

\[
1 \leq IX1 \leq IX2 \leq NDIVIX \\
1 \leq JY1 \leq JY2 \leq NDIVIY
\]

(\(NDIVIX\) and \(NDIVIY\) are specified in RUNSPEC keyword `DIMENS`)

The data should be terminated with a slash (/).

See also the keywords `GDFILE` and `IMPFILE` for fast initialization of the grid and grid arrays by binary read from the full field case.

**Note:** All layers in the Z-direction must be included.

### Example

```plaintext
-- IX1-IX2  JY1-JY2  KZ1-KZ2
SECTOR
10 19 21 40 1 20 /
GRID
-- Read grid
GDFILE
`../FULLFIELD’ /
-- Read grid arrays
IMPFILE
`../FULLFIELD.INIT’ /
PERMX PERMY PERMZ PORO /
```
SEPARATE Requests run summary output to separate RSM file

This keyword requests that the run summary output, generated by using the RUNSUM keyword, should be written to a separate file, rather than be appended to the end of the Print file. The separate run Summary file name is the root with the extension RSM.

It is possible to output the run summaries in a format suitable to be imported into a spreadsheet program such as Lotus 1-2-3 or Excel, by using the LOTUS or EXCEL keyword.

The SEPARATE keyword has no associated data.

See also keyword NARROW.

Example

```
SEPARATE
```
SEPCOND introduces a new separator condition stage

This keyword introduces or modifies a specified separator condition. These separator conditions may then be associated with a well, using keyword WSEPCOND. The separator conditions are used to define oil, gas and liquid rates for a well, by flashing the produced hydrocarbons through the specified separation process.

Each line of SEPCOND data introduces a new stage, specifies the name of the separator condition, and sets the temperature and pressure. Several lines may be used to build up a multi-stage separator condition.

SEPCOND must be used to define a separator condition before it is used by a well, or is given a target.

A separator stage may be re-specified, or the whole separator condition re-specified. The separator stages must, however, be introduced in increasing order, in order to prevent missing stages when the simulation starts.

The default limit on the number of stages per separator is 5. This may be increased using item 5 in the WELLDIMS keyword. The default limit on the number of separators (including the FIELDSEP separator) is 4. This may be increased using item 8 in the WELLDIMS keyword.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. The separator name
   (up to 8 characters).
2. The name of the group for which this separator acts as a default.
   This separator is automatically assigned as the default separator to all wells that can trace their parentage to this group. If a well belonging to this group has a separator explicitly assigned to it using either WSEPCOND or WELLSPEC, it takes precedence over a default separator defined here. If this group contains subgroups, which also have default separators set, then wells belonging to the subgroups take their separator from their parent subgroup and not from this group.

   For a multistage separator, the group name may be entered at any stage; it applies to the whole separator and not just that stage. To avoid confusion, it is best to enter the group name for the first stage only, or for all stages.

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to assign this separator as the default separator to several groups in one record.

   If this item is defaulted or set to a blank (""), the separator is not assigned as a default separator to any group.
3. The stage index.
   DEFAULT: 1
4. The separator stage temperature.
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
   DEFAULT: 15.56 °C (METRIC, LAB and PVT-M), 60 °F (FIELD)
5. The separator stage pressure.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
   DEFAULT: 1.01325 barsa (METRIC), 14.6959 psia (FIELD), 1 atma (LAB and PVT-M)

6. This defines the destination of the liquid output from the separator.

   For all but the last stage in a separator, this is to a later stage, usually the next stage. The default destination is 0, which implies the next stage for all except the last stage, and the stock tank for the last stage.

   Note: If -1 is specified for a stage before the last, this volume is added to the stock tank oil. Care must be taken, as this may result in the accumulation of oil volumes at different pressures or temperatures.

7. This defines the destination stage of the vapor output from the field separator. This is usually accumulated with the stock tank or field separator vapor, the volume rate being taken by conversion to a volume at standard conditions. This is done when the destination stage is set to 0, which is the default.

   In the unusual case in which the vapor output from the field separator is fed to a subsequent stage, a non-default value of the destination stage may be entered. Note that, in this case, if intermediate stage separator gas is used as a source of reinjection gas, ECLIPSE 300 will calculate and report export oil rates. This export rate will be less than the produced oil rate which is the ‘formation’ oil rate prior to any reinjection work. The difference then between the formation oil and export oil is the oil vapor that is contained in the reinjected gas, that would otherwise have been separated in the subsequent separator stage. See also the discussion under "Export tables" in the ECLIPSE Technical Description.

8. The K-value table number.

   This is required if a K-value table rather than an equation of state is being used for the separator. The default value is 0, indicating no K-value table, and implying an equation of state calculation.

9. The gas plant table number.

   This is required if a gas plant table rather than an equation of state is being used for the separator. The default value is 0, indicating no gas plant table, and implying an equation of state calculation.

10. The surface equation of state number.

    This should only be specified if multiple equations of state are used. When this option is employed, it may still be defaulted, in which case the fluid from a well is flashed using the equation of state for the cells in which the well is completed. If an equation of state index is specified, this forces the use of a given surface equation of state. The value must lie between 1 and the maximum number of equations of state for surface conditions, TABDIMS item 10. See also the discussion under "Equations of state regions" in the ECLIPSE Technical Description.

11. Temperature for NGL density evaluation.

    UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

12. Pressure for NGL density evaluation.

    UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
Note: If both items 11 and 12 are specified, an equation of state NGL density calculation is performed, rather than using Amagat's law.

For the usual case, in which a separator chain is connected such that the liquid output from each stage feeds the next, the default values of the destination stages may be used.

Related keywords

For well production, surface volumes are calculated using a well separator, defined by the `WSEPCOND` keyword. If a well separator is not defined, the group separator defined by this keyword `SEPCOND` is used. If the group separator is not defined, the field separator defined by the `FIELDSEP` keyword is used. If the field separator is not defined, a single flash to standard conditions given by `STCOND` is used.

Examples

Example 1

To obtain a three stage separator, the third stage representing the stock tank as shown in figure 3.15:

```
SEPCOND
SEP ' ' 1  80 815  /
SEP ' ' 2  80 65  /
SEP ' ' 3  60 14.7 /
/
```

![Figure 3.15. A three stage separator, with the stock tank as the third stage](image)

Example 2

Use gas plant number 1 to dry all the gas from stages 1, 2 and 3. Liquid from stage 3 is directed to the stock tank oil stream, bypassing the gas plant. The oil rate is the sum of the oil from stages 3 and 4. This separator acts as a default for all wells belonging to group GR1.

```
SEPCOND
SEP GR1 1  85 904  2 4  /
SEP ' ' 2  75 102  3 4  /
SEP ' ' 3  60 14.7  1 4  /
SEP ' ' 4  60 14.7  0 0 1* 1  /
/
```

Keywords

2030
Bo and Rs values from separator test

This keyword is used firstly to define the initial separator conditions in ECLIPSE 100, and subsequently to change them at any time in the simulation. It is required if you wish to model the effects of changing separator conditions in ECLIPSE 100 runs. The first entry of the keyword sets the initial separator conditions. The first entry must be followed by the keyword GSEPCOND, which allocates groups to separators. Subsequent entries of SEPVALS can be used to change the conditions of one or more separators. When the separator conditions have been changed, ECLIPSE transforms the oil and gas flow rates of the wells and groups that feed into the separator. The transformed flow rates are used for reporting and the application of controls and limits.

The keyword can only be used in black oil runs, that is oil must contain dissolved gas, but gas must not contain vaporized oil. There must be at least one TIME, TSTEP or DATES keyword between successive entries of SEPVALS, in order to register the change in conditions.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Name of the separator
   Up to 8 characters.
   There can be up to NGMAX2 separators, where NGMAX2 is the maximum number of groups, entered using keyword WELLDIMS in RUNSPEC.
   The separator must be allocated to a group with keyword GSEPCOND.
   Separator name roots are not allowed. An error message is issued if the name contains an asterisk (*).

2. Formation volume factor (B_{ob}) of bubble point oil, when flashed from reservoir conditions (P_{bub}, T_{res}) to stock tank conditions through the separator system.
   UNITS: rm^3/sm^3 (METRIC), rb/stb (FIELD), rcc/scc (LAB), rm^3/sm^3 (PVT-M)

3. Solution gas-oil ratio (R_{si}) of bubble point oil, when flashed from reservoir conditions (P_{bub}, T_{re}) to stock tank conditions through the separator system.
   UNITS: sm^3/sm^3 (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm^3/sm^3 (PVT-M)

   End the data record with a slash (/).

   End the set of records with a blank record, containing just a slash (/).

See also keywords GSEPCOND and COMPFISH.

Example

Set initial separator conditions:

```
SEPVALS
  -- separator name   Bob   Rsi
   SEP1     1.251  0.510 /
   SEP2     1.261  0.515 /
/
```

Assign separators to groups:

```
GSEPCOND
  -- group name   separator name
```

Keywords
Simulate:

\[
\text{TIME or TSTEP or DATES}
\]

Change conditions of SEP1:

\[
\text{SEPVALS}
\]

Continue simulation:

\[
\text{TIME or TSTEP or DATES}
\]

### Separator tests

PVT analysis of a sample of reservoir fluid normally consists of three parts:

- A determination of the bubble point pressure, \( P_{\text{bub}} \).
- A differential liberation, to determine the change in fluid properties and the amount of gas liberated as the reservoir pressure declines from \( P_{\text{bub}} \).
- A separator test, in which the fluid sample is flashed from bubble point conditions (\( P_{\text{bub}}, T_{\text{res}} \)) to stock tank conditions through the separator system, at particular sets of separator conditions.

The separator test yields the two items required in this keyword. The formation volume factor (\( B_{\text{ob}} \)) from the test is the ratio of the volume of oil at bubble point conditions to the volume of oil that comes through to the stock tank. (Its reciprocal is called the shrinkage factor.) The solution gas-oil ratio (\( R_{\text{si}} \)) is the volume of gas, at standard conditions, that was removed by the separator system from a unit volume of oil in the stock tank.

The data entered in the \texttt{PVTO} keyword is prepared by a combined PVT analysis of the differential liberation and separator test results. Thus the values in the \texttt{PVTO} tables are specific to a particular set of separator conditions. ECLIPSE assumes that the \texttt{PVTO} tables were prepared using the original separator conditions, and it is up to you to ensure that the first entry of \texttt{SEPVALS} contains the values of \( B_{\text{ob}} \) and \( R_{\text{si}} \) from the separator test that were used to prepare the tables.

### How ECLIPSE 100 handles a change of separator conditions

A change in separator conditions alters the relationship between fluid volumes at reservoir conditions and stock tank conditions. The differential liberation data (which governs the shape of the \texttt{PVTO} table) will of course remain the same. The effect of a separator change is to multiply the oil formation volume factors by a constant factor, and to change the \( R_{\text{s}} \) values by a certain amount.

In fact, ECLIPSE does not transform the \texttt{PVTO} tables. All calculations of fluid flow and material balance within the reservoir are performed using the \texttt{PVTO} tables as input by the engineer. However, the oil and gas flows through the wells are transformed to reflect the change in separator conditions.
Let the original separator test results be $B_{ob}$ and $R_{si}$, and the test results using the new separator conditions be $B_{ob}^*$ and $R_{si}^*$. If $Q_o$ and $Q_g$ are the oil and gas flow rates at the original separator conditions (that is using the PVTO tables as input), then the oil and gas flow rates at the new separator conditions will be:

$$Q_o^* = T_{oil-oil} Q_o$$  \hspace{1cm} \text{Eq. 3.168}$$

$$Q_g^* = T_{gas-oil} Q_o + Q_g$$  \hspace{1cm} \text{Eq. 3.169}$$

Where the transformation coefficients are:

$$T_{oil-oil} = \frac{B_{ob}}{B_{ob}^*}$$  \hspace{1cm} \text{Eq. 3.170}$$

$$T_{gas-oil} = T_{oil-oil} R_{si}^* - R_{si}$$  \hspace{1cm} \text{Eq. 3.171}$$

After a change in separator conditions, the transformation coefficients is printed in a table of separator data if you request output of the well scheduling data (RPTSCHED mnemonic WELSPECS or switch 14 > 0).

The well and group flow rates are transformed for output. Well and group flow targets and limits, and economic limits, are applied to the transformed rates. The fluids-in-place reports and inter-region flows, however, will still be reported at the original separator conditions.

In tubing head pressure calculations, the flow rates used to look up the VFP tables are not transformed to the new separator conditions. Thus if the VFP tables were prepared using PVT data at the original separator conditions, you will not have to recalculate them after a change in separator conditions. As a consequence, however, the pressure drops will no longer match the reported flows, since the reported flows are transformed to the new separator conditions. A similar policy is applied to pipeline pressure drops when using the Network option. In any pipeline subordinate to a separator with changed conditions, the flows are back-transformed to the original conditions for the pressure drop calculation.
Initial foam concentrations

This keyword sets the initial foam concentration in runs where the initial conditions are specified by enumeration.

The keyword should be followed by one real number for every grid block specifying the initial foam concentration. The data field should be terminated by a slash (/).

If item 1 of FOAMOPTS is set to ‘GAS’:

UNITS: kg/sm$^3$ (METRIC), lb/Mscf (FIELD), gm/scc (LAB)

If item 1 of FOAMOPTS is set to ‘WATER’:

UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scc (LAB)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*0.18). Note that spaces must not be inserted on either side of the asterisk (*).

See also the keywords PRESSURE, SGAS, RS, PBUB, RV, PDEW and SWAT in the SOLUTION section.

Refer to "Foam model" in the ECLIPSE Technical Description for further information.

Note: When adsorption is active, the concentration specified here will be honored and extra adsorbed foam is added to the rock so that it is in equilibrium with the input fluid concentration.

Note: If a non-zero concentration is specified in cells containing none of the host phase, the concentration will be automatically reset to zero. This behavior can be turned off using item 165 in OPTIONS.

Example

With NDIVIX=10, NDIVIY=1 and NDIVIZ=2 specified in RUNSPEC keyword DIMENS

```
SFOAM
10*0.0
9*0.0 0.5 /
```
The keyword should be followed by one real number for every grid block specifying the initial gas saturation. The data field should be terminated by a slash (/).

This keyword may be used to specify the initial solution explicitly, as an alternative to equilibration using EQUIL. It is expected if the PRESSURE keyword is present, indicating explicit initialization.

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*0.18). Note that spaces must not be inserted on either side of the asterisk (*).

See also the keywords PRVD, RS, PBUB, RV, PDEW and SWAT in the SOLUTION section.

In ECLIPSE 300, the same information may be obtained from a restart file using the GETSOL keyword.

For the ECLIPSE Thermal model using the mixed hydrostatic equilibration option, the initial saturations should be specified together with items 1 and 2 of the EQUIL keyword and the NEI keyword. For more information, refer to "Initializing the study" in the ECLIPSE Technical Description.

When using explicit initialization, you must ensure that the initial solution is stable and physically reasonable.

Example

With NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS:

```
SGAS
48*0.4
48*0.04
48*0
48*0 /
```
SGCR

**Scaled critical gas saturations**

The SGCR and ISGCR keywords specify the critical gas saturation (that is the largest gas saturation for which the gas relative permeability is zero) within each grid cell. The free gas flow across each grid face is calculated from a transformed gas relative permeability curve obtained by linearly scaling the tabulated relative permeability data between the new critical gas saturation defined using an SGCR keyword, and the new maximum gas saturation (see under keyword SGU in the PROPS section). The SGCR keyword also permits scaling of the relative permeability table used in computing the flow of free gas between grid cells and well connections.

Seven keywords are provided to allow the critical gas saturation for each grid cell to be scaled independently.

These keywords are:

- **SGCR**: Critical gas saturation for the non-directional saturation function tables
- **SGCRX**: Critical gas saturation for the +X face of each grid cell
- **SGCRX−**: Critical gas saturation for the -X face of each grid cell
- **SGCRY**: Critical gas saturation for the +Y face of each grid cell
- **SGCRY−**: Critical gas saturation for the -Y face of each grid cell
- **SGCRZ**: Critical gas saturation for the +Z face of each grid cell
- **SGCRZ−**: Critical gas saturation for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the critical gas saturations for flows in the +R,-R,+T,-T,+Z,-Z directions respectively.

None of the SGCR keywords should be used if gas is not present in the model. The keyword ENDSCALE should be specified in the RUNSPEC section.

If both directional and irreversible end point scaling is requested, using switches 'DIRECT' and 'IRREVERS' in the ENDSCALE keyword in RUNSPEC, then the critical gas saturations for gas flow across the +X and -X faces of a grid cell are set using the keywords SGCRX, SGCRX−. Similarly SGCRY, SGCRY− are used for the +Y,-Y faces and the keywords SGCRZ, SGCRZ− for the +Z,-Z faces. Scaling of the well connection relative permeability tables for any well connections in the grid cell are performed using the SGCR keyword.

When the end point scaling is requested to be directional and reversible, using the switch 'DIRECT' but not 'IRREVERS' in ENDSCALE, the keyword SGCRX defines the critical gas saturations for the +X and -X faces of each grid cell. Keywords SGCRY, SGCRZ are used for both the +Y, -Y faces and the +Z, -Z faces of each grid cell respectively. Scaling of well connection relative permeability tables uses data entered under the SGCR keyword.

If ENDSCALE is specified in RUNSPEC without the switch 'DIRECT', then none of the directional forms of the SGCR keyword are allowed. The SGCR keyword then defines the new critical gas saturation for flow out of each face of the grid cell and for gas flows between the grid cell and well connections.

Each SGCR keyword should be followed by one real number for each grid block in the current input box specifying the critical gas saturation within the grid block. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash(/).
Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If any of the keywords SGCR, SGCRX, SGCRX-, SGCRY, SGCRY-, SGCRZ, SGCRZ- are omitted in a model where they could be accepted then the critical gas saturation for a gas relative permeability calculation defaults to the value used in the appropriate relative permeability table.

If the Hysteresis model (switch 'HYSTER' in keyword SATOPTS) is active then the SGCR data will scale the drainage curve. The imbibition curve can be scaled by using the equivalent ISGCR (ISGCRX, ISGCRX-, ...) keyword.

If the ODD3P relative permeability and capillary pressure hysteresis model has been selected, the primary end-point scaling is specified using the PSORG, PSGRO, PSORW, PSWRO, PSGRW and PSWRG keywords and the hysteresis end-point scaling is specified using the HSORG, HSGRO, HSORW, HSWRO, HSGRW and HSWRG keywords

See also keywords SGU, SGL, ENPTVD in the PROPS section.

For details of the end point rescaling procedure, refer to "Saturation table scaling" in the ECLIPSE Technical Description.

Example

```eclipse
-----RUNSPEC section
DIMENS
  1 7 3 /
ENDSCALE
/
-----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3
SGCR
  5*0.04 10*0.03 3*0.05 /
```
Miscible critical gas saturation tables

The data comprises NTMISC (item 1 in keyword MISCIBLE) tables of miscible critical gas saturation versus water saturation, each terminated by a slash. See "Miscible flood modeling" in the ECLIPSE Technical Description for details of how the effective critical gas saturation tables are used in the calculation of the gas and oil relative permeabilities and PVT properties. See also "Solvent model" in the ECLIPSE Technical Description for further information on when the four-component solvent model is employed.

The SGCWMIS keyword can only be used in runs which use the miscible flood option (see keyword MISCIBLE) and where oil, water and gas are all present. For miscible flood simulations involving the water phase the SGCWMIS keyword may be omitted, in which case the program assumes a miscible critical gas saturation of zero.

Each table consists of 2 columns of data:

1. The water saturation.
   - Values should be between 0 and 1 and should increase monotonically down the column.

2. The corresponding miscible critical gas saturation.
   - Values should be between 0 and 1 and should be level or increasing down the column.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSMISC (item 2 in keyword MISCIBLE).

Example

With NTMISC=2 and NSMISC≥3

<table>
<thead>
<tr>
<th>SGCWMIS</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>.00 .00</td>
<td></td>
</tr>
<tr>
<td>1.00 .00</td>
<td>/</td>
</tr>
<tr>
<td>.00 .00</td>
<td></td>
</tr>
<tr>
<td>.25 .05</td>
<td></td>
</tr>
<tr>
<td>1.00 .05</td>
<td>/</td>
</tr>
</tbody>
</table>
SGF3

Three-phase gas saturation functions

This keyword should only be used when the IKU three-phase relative permeability model has been requested using the  **IKU3P** keyword or when the ODD3P three-phase relative permeability and capillary pressure hysteresis model has been requested using the  **ODD3P** keyword or when the gas BAKER three-phase relative permeability and capillary pressure hysteresis model has been requested using the  **BAKER1** or  **BAKER2** keywords in the  **PROPS** section.

The data specified consists of tables of two-phase gas saturation functions, each terminated by a slash. Each table consists of 4 columns of data:

Column:

1. The gas saturation.
   Values should be between 0 and 1 and should increase monotonically down the column.

2. The corresponding gas in oil relative permeability measured in the absence of water.
   Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0.

3. The corresponding gas in water relative permeability measured in the absence of oil.
   Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0.

   For the  **ODD3P** option the relative permeability values between the critical and maximum saturations are not used and may be defaulted.

4. The corresponding oil-gas capillary pressure.
   Values should be level or increasing down the column.

   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

The number of tables supplied should be equal to the number of saturation tables in the run, specified in item 1 of keyword  **TABDIMS** in the  **RUNSPEC** section.

The number of rows should be less than or equal to the maximum number of rows of saturation data specified in item 3 of  **TABDIMS**.

See also "Saturation functions" in the  **ECLIPSE Technical Description**.

**Note:** The critical saturation for each relative permeability function is established by determining the highest saturation in the table for which the relative permeability is notionally zero, that is, less than a small threshold value. Relative permeability values which are lower than this threshold value will be treated as zero. For ECLIPSE 100 and ECLIPSE 300 the default threshold values are 1.0 x 10^{-6} and 1.0 x 10^{-20} respectively. These values may be modified using the  **TOLCRIT** keyword.

### Example

```
SGF3
.00  .00  .00  .0
.04  .00  .00  .2
.10  .00  .022 .5
.20  .05  .10  1.0
.40  .15  .34  2.0
```
<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>.60</td>
<td>.30</td>
<td>.50</td>
<td>3.0</td>
</tr>
<tr>
<td>.78</td>
<td>1.00</td>
<td>1.00</td>
<td>3.9</td>
</tr>
</tbody>
</table>
This keyword is an alternative to keyword SGFN, allowing the three-phase relative permeability of gas to be input directly as a function of oil and water saturations in a two-dimensional table.

The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) tables of gas relative permeability, tabulated against both oil and water saturations, each terminated by a slash (/).

Each table consists of a number of records:

Record 1: The oil saturation values

Values should be between 0 and 1 and should increase monotonically. The number of oil saturation values should not exceed NSSFUN (see item 3 of keyword TABDIMS).

Subsequent records (a maximum of NSSFUN records):

The water saturation.

Followed by the gas relative permeability values for this water saturation at all the corresponding oil saturations entered in Record 1.

The gas relative permeabilities should be monotonically decreasing, with a final value of zero. The number of gas relative permeability values should not exceed the number of oil saturation values entered in Record 1. The record may be truncated at any point provided that the last relative permeability value is zero.

The columns of data must also comply with the following constraints:

• The water saturation (the first item in the second and subsequent records) should increase monotonically down the column. The gas relative permeability for each oil saturation should decrease monotonically down the column and should end in a zero value.

• The relative permeability must also be zero for any point on the table in which \( S_{oil} + S_{wat} \geq 1.0 \).

• Default values may not be placed within the table.

The entire table may be defaulted provided that the table is not the first. Defaulted tables are replaced with a copy of the previous table.

The gas/oil capillary pressure must be provided using the PCG32D keyword.

The gas/oil capillary pressure can be provided using the SGFN keyword. The relative permeability data entered with the SGFN keyword will be ignored.

The keyword cannot be used with the end-point scaling, hysteresis and miscible options.

Example

With NTSFUN=1 and NSSFUN \( \geq 11 \):

```
SGF32D
----------------------------------------------------------SOIL----------------------------------------------------------
| 0.01 | 0.27 | 0.32 | 0.37 | 0.42 | 0.47 | 0.52 | 0.57 | 0.72 | 0.77 | 0.78 | / |
---SWAT---
0.22 1.000 0.625 0.345 0.207 0.113 0.083 0.053 0.023 0.002 0.001 0.000/ |
0.25 0.555 0.337 0.210 0.110 0.078 0.047 0.021 0.004 0.001 0.000/ |
0.30 0.330 0.212 0.106 0.074 0.042 0.019 0.003 0.002 0.000/ |
0.35 0.215 0.103 0.069 0.036 0.017 0.003 0.002 0.001 0.000/ |
```
<table>
<thead>
<tr>
<th></th>
<th>0.40</th>
<th>0.100</th>
<th>0.065</th>
<th>0.031</th>
<th>0.015</th>
<th>0.002</th>
<th>0.002</th>
<th>0.001</th>
<th>0.000</th>
<th>/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.50</td>
<td>0.060</td>
<td>0.025</td>
<td>0.014</td>
<td>0.002</td>
<td>0.001</td>
<td>0.001</td>
<td>0.000</td>
<td>/</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.65</td>
<td>0.020</td>
<td>0.012</td>
<td>0.001</td>
<td>0.001</td>
<td>0.001</td>
<td>0.000</td>
<td>/</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.80</td>
<td>0.010</td>
<td>0.001</td>
<td>0.001</td>
<td>0.001</td>
<td>0.000</td>
<td>/</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>1.00</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>/</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Gas saturation functions

The data comprises `NTSFUN` (see item 1 of keyword `TABDIMS` in the `RUNSPEC` section) tables of gas saturation functions, each terminated by a slash (/).

Each table consists of 3 columns of data:

1. The gas saturation.
   Values should be between 0 and 1 and should increase monotonically down the column.

2. The corresponding gas relative permeability.
   Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0.

3. The corresponding oil-gas capillary pressure.
   Values should be level or increasing down the column.
   
   **UNITS:** bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

   In gas-water systems (that is no active oil phase), the gas-water capillary pressure is entered in the `SWFN` keyword. The oil-gas capillary pressure in `SGFN` should be set to zero.

   Default values (represented by 1*) may be inserted as required in columns 2 and 3. When the table is read in, defaults are replaced by values computed by linear interpolation.

   There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than `NSSFUN` (see item 3 of keyword `TABDIMS`).

   The entire table may be defaulted provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

   The keyword families:

   (i) `SWOF, SGOF, SLGOF`

   (ii) `SWFN, SGFN, SGWFN, SOF2, SOF3, SOF32D`

   provide two alternative formats for inputting the saturation functions. Do not mix keywords from the two families in the same run. In gas-water runs, or if you are using the ECLIPSE 100 miscible flood or solvent options, the saturation functions must be input with keywords from family (ii).

   In gas-water runs the gas and water saturation functions may be input together using the `SGWFN` keyword as an alternative to using both `SGFN` and `SWFN`.

   **ECLIPSE 300**

   Any non-zero connate gas saturation entered with this keyword will be ignored, that is reset to zero.

   **Note:** The critical saturation for each relative permeability function is established by determining the highest saturation in the table for which the relative permeability is notionally zero, that is, less than a small threshold value. Relative permeability values which are lower than this threshold value will be treated as zero. For ECLIPSE 100 and ECLIPSE 300 the default threshold values are $1.0 \times 10^{-6}$ and $1.0 \times 10^{-20}$ respectively. These values may be modified using the `TOLCRIT` keyword.
**Example**

With `NTSFUN=2` and `NSSFUN ≥ 9`:

```plaintext
SGFN
-0.000 0.000 0.000
-0.040 0.000 0.2000
-0.100 0.0220 0.5000
-0.200 0.1000 1.0000
-0.300 0.2400 1*
-0.5000 0.4200 1*
-0.6000 0.5000 3.0000
-0.7000 0.8125 3.5000
-0.7800 1.0000 3.9000
/ .00 0.000 0
-0.12 0.0220 0
-0.22 0.1000 0
-0.32 0.2400 0
-0.52 0.4200 0
-0.62 0.5000 0
-0.72 0.8125 0
-0.80 1.0000 0
/```

```
Keywords

2044
Scaled connate gas saturations

The different forms of this keyword specify the connate gas saturation (that is the smallest gas saturation in a gas saturation function table) within each grid cell. The gas-oil capillary pressure for gas flow across each grid face is calculated from a transformed gas-oil capillary pressure curve obtained by linearly scaling the tabulated capillary pressure data between the new connate gas saturation defined using an SGL keyword, and the new maximum gas saturation (see under the keyword SGU in the PROPS section). The SGL keyword also permits scaling of the gas-oil capillary pressure table used in computing the initial gas saturations.

Seven keywords are provided to allow the connate gas saturation for each grid cell to be scaled independently. These keywords are:

- **SGL** : Connate gas saturation for the non-directional saturation function tables
- **SGLX** : Connate gas saturation for the +X face of each grid cell
- **SGLX-** : Connate gas saturation for the -X face of each grid cell
- **SGLY** : Connate gas saturation for the +Y face of each grid cell
- **SGLY-** : Connate gas saturation for the -Y face of each grid cell
- **SGLZ** : Connate gas saturation for the +Z face of each grid cell
- **SGLZ-** : Connate gas saturation for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the connate gas saturations for flows in the +R,-R,+T,-T,+Z,-Z directions respectively.

None of the SGL keywords should be used if gas is not present in the model. The keyword **ENDSCALE** should be specified in the RUNSPEC section.

If both directional and irreversible end point scaling is requested, using switches 'DIRECT' and 'IRREVERS' in the **ENDSCALE** keyword in RUNSPEC, then the connate gas saturations for gas flow across the +X and -X faces of a grid cell are set using the keywords SGLX, SGLX-. Similarly SGLY, SGLY- are used for the +Y,-Y faces and the keywords SGLZ, SGLZ- for the +Z,-Z faces. Scaling of the gas-oil capillary pressure tables used in the equilibration algorithm are performed with the SGL keyword.

When the end point scaling is requested to be directional and reversible, using the switch 'DIRECT' but not 'IRREVERS' in **ENDSCALE**, the keyword SGLX defines the connate gas saturations for the +X and -X faces of each grid cell. Keywords SGLY, SGLZ are used for both the +Y, -Y faces and the +Z,-Z faces of each grid cell respectively. Scaling of gas-oil capillary pressure data for the equilibration algorithm is done with the SGL keyword.

If **ENDSCALE** is specified in RUNSPEC without the switch 'DIRECT', then none of the directional forms of the SGL keyword are allowed. The SGL keyword then defines the new connate gas saturation for scaling the gas flow out of each grid cell face and for calculating the transformed capillary pressure table used in the equilibration procedure.

Each SGL keyword should be followed by one real number for each grid block in the current input box specifying the connate gas saturation within the grid block. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash(/).
Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If any of the keywords SGL, SGLX, SGLX-, SGLY, SGLY-, SGLZ, SGLZ- are omitted in a model where they could be accepted then the connate gas saturation for a gas-oil capillary pressure calculation defaults to the value used in the appropriate gas saturation function table.

If a connate gas saturation is entered using the SGLPC keyword then this saturation will be used when scaling the capillary pressure curves. The connate gas saturation entered using SGL will only be used to scale the oil relative permeability curves.

If the Hysteresis Model (switch 'HYSTER' in keyword SATOPTS) is active then the SGL data will scale the drainage curve. The imbibition curve can be scaled by using the equivalent ISGL (ISGLX, ISGLX-, ...) keyword.

**Note:** In some cases, where messages relating to end-point scaling are emitted, for example, when checking that the scaled end-points are consistent, reference is made to SGCO as an alternative to SGL.

---

**ECLIPSE 100**

In ECLIPSE 100 the connate gas saturation is not included in the two-point end-point scaling of oil relative permeability to gas and water by default. In order to include connate gas in the maximum attainable gas saturation calculation, item 205 in the OPTIONS keyword should be set to 1.

**ECLIPSE 300**

In ECLIPSE 300 the unscaled connate gas saturation is set to zero by default and this overrides any non-zero values in the input saturation tables. However, non-zero unscaled connate gas saturations can be modeled by setting item 103 of the OPTIONS3 keyword to 1. Non-zero scaled connate gas saturations may be specified by the SGL keyword.

See also keywords SGLPC, SGU and ENPTVD in the PROPS section.

See "Saturation table scaling" in the ECLIPSE Technical Description for further details.

---

**Example**

```plaintext
----RUNSPEC section
DIMNS 1 7 3 /
ENDSCALE
/
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX 1 1 2 7 1 3
SGL 5*0.0 10*0.02 3*0.0 /
```

---

**Keywords**

2046
Scaled connate gas saturations, for \(P_c\) curves only

The SGLPC and ISGLPC keywords scale the connate gas end point of the gas-oil capillary pressure curves without scaling the corresponding oil relative permeability curves.

The SGLPC keyword should not be used unless gas is present in the model (keyword \texttt{GAS} in the RUNSPEC section). The keyword \texttt{ENDSCALE} should be specified in the RUNSPEC section.

The SGLPC keyword should be followed by one real number for each grid block in the current input box, specifying the connate gas saturation within the grid block. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If SGLPC is not specified, the gas-oil capillary pressure scaling is performed using the connate gas saturation specified with the SGL family of keywords (SGL, SGLX, SGLX-, SGLY, SGLY-, SGLZ, SGLZ-). If in turn any of the keywords SGL, SGLX, SGLX-, SGLY, SGLY-, SGLZ, SGLZ- are omitted, then the connate gas saturation for a gas-oil capillary pressure calculation defaults to the value used in the appropriate gas saturation function table.

If the hysteresis model (switch 'HYSTER' in keyword \texttt{SATOPTS}) is active then the SGLPC data will scale the drainage curve. The imbibition curve can be scaled by using the equivalent ISGLPC keyword.

See also keywords SGL and ENPTVD in the PROPS section.

Example

With ENDSCALE in RUNSPEC; and NDIVIX=1, NDIVIY=7 and NDIVIZ=3 specified in RUNSPEC keyword \texttt{DIMENS}:

```
SGLPC
7*0.0 7*0.02 7*0.0 /
```
**Gas/oil saturation functions versus gas saturation**

The **SGOF** keyword may be used in runs containing both oil and gas as active phases, to input tables of gas relative permeability, oil-in-gas relative permeability and oil-gas capillary pressure as functions of the gas saturation. If water is also an active phase in the run, the water/oil saturation functions must be input with keyword **SWOF**.

The data comprises **NTSFUN** (see item 1 of keyword **TABDIMS** in the RUNSPEC section) tables of gas/oil saturation functions, each terminated by a slash (/).

Each table consists of 4 columns of data:

1. **The gas saturation.**
   - Values should be between 0 and 1 and should increase monotonically down the column. The first value in the column must be zero.

2. **The corresponding gas relative permeability.**
   - Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be zero.

3. **The corresponding oil relative permeability when oil, gas and connate water are present.**
   - Values should be between 0 and 1 and should be level or decreasing down the column. If water is present in the run, the first value in the column (\(k_{row} \text{ at } S_g = 0\)) must be the same as the first value in the krow column in keyword **SWOF** (that is \(k_{row} \text{ at } S_o = 1 - S_{wco}\)). The last value in the column must be zero.

4. **The corresponding oil-gas capillary pressure.**
   - Values should be level or increasing down the column.

   **UNITS:** bars (METRIC), psi (FIELD), atm (LAB), atm (PVT–M)

Default values (represented by 1*) may be inserted as required in columns 2, 3 and 4. When the table is read in, defaults are replaced by values computed by linear interpolation.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than **NSSFUN** (see item 3 of keyword **TABDIMS**).

The entire table may be defaulted provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

The keyword families

(i) **SWOF, SGOF, SLGOF**

(ii) **SWFN, SGFN, SGWFN, SOF2** and **SOF3**

provide two alternative formats for inputting the saturation functions. Do not mix keywords from the two families in the same run. In gas-water runs, or if you are using the ECLIPSE 100 miscible flood or solvent options, the saturation functions must be input with keywords from family (ii).

For two dimensional tables the following keywords can also be used: **SOF32D, SGF32D** and **SWP32D**.

---

**Keywords**
Note: The critical saturation for each relative permeability function is established by determining the highest saturation in the table for which the relative permeability is notionally zero, that is, less than a small threshold value. Relative permeability values which are lower than this threshold value will be treated as zero. For ECLIPSE 100 and ECLIPSE 300 the default threshold values are $1.0 \times 10^{-6}$ and $1.0 \times 10^{-20}$ respectively. These values may be modified using the `TOLCRIT` keyword.

See "Saturation functions" in the ECLIPSE Technical Description for more information, and a description of the methods used to compute three-phase relative permeabilities.

Example

With `NTSFUN=2` and `NSSFUN \geq 10`:

```
  SGOF
  -- Sg   Krg   Krog   Pcog
     .0000   .0000   1.00     .0000
     .0400   .0000   0.60     .2000
     .1000   .0220   0.33     .5000
     .2000   .1000   0.10     1.0000
     .3000   .2400   0.02     1*
     .4000   1*     0.00     1*
     .5000   .4200   0.00     1*
     .6000   .5000   0.00     3.0000
     .7000   .8125   0.00     3.5000
     .7800   1.0000   0.00     3.9000   / table 1
     .82   1.0000   0.00     0
     .95   .2220   0.62     0
     .2   .1100   0.12     0
     .5   .4400   0.0     0
     .6   .5600   0.0     0
     .82   1.0000   0.0     0   / table 2
```

Keywords

2049
Scaled saturation table maximum gas saturations

The different forms of this keyword specify the maximum gas saturation (that is the largest gas saturation in a gas saturation function table) within each grid cell. The scaled maximum gas saturation is used to determine the scaled forms for the gas-oil capillary pressure curves and the gas relative permeability curves. The scaled form of the capillary pressure curves is obtained by linearly scaling the tabulated capillary pressure data between the new connate gas saturation (see keyword SGL in the PROPS section) and the new maximum gas saturation defined using an SGU keyword. A similar linear transformation is used to produce the scaled gas relative permeability curves using the new critical gas saturation (see keyword SGCR in the PROPS section) and the new maximum gas saturation defined using an SGU keyword. The scaled capillary pressure and relative permeability curves are then used to compute the free gas flow out of each grid cell face, the free gas flow between grid cells and well connections and in the equilibration algorithm.

Seven keywords are provided to allow the maximum gas saturation for each grid cell to be scaled independently. These keywords are:

- **SGU**: Maximum gas saturation for the non-directional saturation function tables
- **SGUX**: Maximum gas saturation for the +X face of each grid cell
- **SGUX-**: Maximum gas saturation for the -X face of each grid cell
- **SGUY**: Maximum gas saturation for the +Y face of each grid cell
- **SGUY-**: Maximum gas saturation for the -Y face of each grid cell
- **SGUZ**: Maximum gas saturation for the +Z face of each grid cell
- **SGUZ-**: Maximum gas saturation for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the maximum gas saturations for flows in the +R,-R,+T,-T,+Z,-Z directions respectively.

None of the SGU keywords should be used if gas is not present in the model. The keyword ENDSCALE should be specified in the RUNSPEC section.

If both directional and irreversible end-point scaling is requested, using switches 'DIRECT' and 'IRREVERS' in the ENDSCALE keyword, then the maximum gas saturations for computing the gas flow across the +X and -X faces of a grid cell are set using the keywords SGUX, SGUX-. Similarly SGUY, SGUY- are used for the +Y, -Y faces and the keywords SGUZ, SGUZ- for the +Z, -Z faces. Scaling of the gas-oil capillary pressure tables in the equilibration algorithm and scaling of the relative permeability curves to compute free gas flows between grid cells and well connections are performed using the SGU keyword.

When the end-point scaling is requested to be directional and reversible, using the switch 'DIRECT' but not 'IRREVERS' in ENDSCALE, the keyword >SGUX defines the maximum gas saturations for the +X and -X faces of each grid cell. Keywords SGUY, SGUZ are used for both the +Y, -Y faces and the +Z, -Z faces of each grid cell respectively. The maximum gas saturation values for use in the equilibration procedure and for computing the well connection flows are entered using the SGU keyword.

If ENDSCALE is specified in RUNSPEC without the switch 'DIRECT', then none of the directional forms of the SGU keyword are allowed. The SGU keyword then defines the new maximum gas saturation for
computing the gas flows between adjacent grid cells, well connection flows and in the equilibration algorithm.

Each SGU keyword should be followed by one real number for each grid block in the current input box specifying the maximum gas saturation within the grid cell. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If any of the keywords SGU, SGUX, SGUX-, SGUY, SGUY-, SGUZ, SGUZ- are omitted in a model where they could be accepted then the maximum gas saturation for capillary pressure and relative permeability calculations defaults to the value used in the appropriate gas saturation function table.

If the hysteresis model (switch 'HYSTER' in keyword SATOPTS) is active then the SGU data will scale the drainage curve. The imbibition curve can be scaled by using the equivalent ISGU (ISGUX, ISGUX-,....) keyword.

See also keywords SGL, SGCR, ENPTVD in the PROPS section.

**Example**

```plaintext
-----RUNSPEC section
DIMENS 1 7 3 /
ENDSCALE /
-----PROPS section
---------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX 1 1 2 7 1 3
SGU 18*1.00 /
```

**Keywords**

2051
Scaled critical gas saturations in water

This keyword specifies the critical gas saturation in water within each grid cell. This is available when the \texttt{IKU3P}, \texttt{BAKER1} or \texttt{BAKER2} three-phase relative permeability model is being used and the two-phase gas-water relative permeabilities are specified using the \texttt{SGF3} keyword. The critical gas saturation in oil is obtained from \texttt{SGCR}.

Seven keywords are provided to allow the critical gas saturation in water for each grid cell to be scaled independently. These keywords are:

- \texttt{SGWCR}  
  Critical gas saturation in water for the non-directional saturation function tables

- \texttt{SGWCRX}  
  Critical gas saturation in water for the \(+X\) face of each grid cell

- \texttt{SGWCRX-}  
  Critical gas saturation in water for the \(-X\) face of each grid cell

- \texttt{SGWCRY}  
  Critical gas saturation in water for the \(+Y\) face of each grid cell

- \texttt{SGWCRY-}  
  Critical gas saturation in water for the \(-Y\) face of each grid cell

- \texttt{SGWCRZ}  
  Critical gas saturation in water for the \(+Z\) face of each grid cell

- \texttt{SGWCRZ-}  
  Critical gas saturation in water for the \(-Z\) face of each grid cell

In radial grids the same directional keywords should be used to represent the critical gas saturations in water for flows in the \(+R\), \(-R\), \(+T\), \(-T\), \(+Z\) and \(-Z\) directions respectively.

None of the \texttt{SGWCR} keywords should be used if water is not present in the model. The keyword \texttt{ENDSCALE} should be specified in the \texttt{RUNSPEC} section.

If both directional and irreversible end-point scaling are requested, using switches \texttt{'DIRECT'} and \texttt{'IRREVERS'} in the \texttt{ENDSCALE} keyword in \texttt{RUNSPEC}, the critical gas saturations in water for flow across the \(+X\) and \(-X\) faces of a grid cell are set using the keywords \texttt{SGWCRX} and \texttt{SGWCRX-}. Similarly the keywords \texttt{SGWCRY} and \texttt{SGWCRY-} are used for the \(+Y\) and \(-Y\) faces and the keywords \texttt{SGWCRZ} and \texttt{SGWCRZ-} for the \(+Z\) and \(-Z\) faces respectively. Scaling of the well connection relative permeability tables for any well connections in the grid cell is performed using the values specified via the \texttt{SGWCR} keyword.

When the end-point scaling is requested to be directional and reversible, using the switch \texttt{'DIRECT'} but not \texttt{'IRREVERS'} in \texttt{ENDSCALE}, the keyword \texttt{SGWCRX} defines the critical gas saturations in water for the \(+X\) and \(-X\) faces of each grid cell. Keywords \texttt{SGWCRY} and \texttt{SGWCRZ} are used for both the \(+Y\) and \(-Y\) faces and the \(+Z\) and \(-Z\) faces of each grid cell respectively. Scaling of the well connection relative permeability tables for any well connections in the grid cell is performed using the values specified via the \texttt{SGWCR} keyword.

If \texttt{ENDSCALE} is specified in \texttt{RUNSPEC} without the switch \texttt{'DIRECT'}, none of the directional forms of the \texttt{SGWCR} keyword are allowed. The \texttt{SGWCR} keyword then defines the critical gas saturation in water for flow out of each face of the grid cell and for flows between the grid cell and well connections.

Each \texttt{SGWCR} keyword should be followed by one real number for each grid block in the current input box specifying the critical gas saturation in water within the grid block. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.
If any of the keywords SGWCR, SGWCRX, SGWCRX-, SGWCRY, SGWCRY-, SGWCRZ or SGWCRZ- are omitted in a model where they could be accepted, the critical gas saturation in water for a gas relative permeability calculation defaults to the value specified in the appropriate relative permeability table.

If the hysteresis model (switch 'HYSTER' in keyword SATOPTS) is active, the SGWCR data scales the drainage curve. The imbibition curve can be scaled by using the corresponding ISGWCR (ISGWCRX, ISGWCRX-, ...) keywords. (The IKU3P three phase relative permeability model does not support hysteresis.)

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```
----RUNSPEC section
DIMENS
  1 7 3 /
ENDSCALE

----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3 /
SGWCR
  5*0.2 10*0.1 3*0.1 /
ENDBOX
```
Gas-water saturation functions

The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) tables of gas-water saturation functions, each terminated by a slash (/).

The keyword may be used in gas-water systems (when no oil is present) as a convenient way of providing saturation function data for both gas and water, as an alternative to using the SGFN and SWFN keywords.

Each table consists of 4 columns of data:

1. The gas saturation.
   Values should be between 0 and 1 and should increase monotonically down the column.
2. The corresponding gas relative permeability.
   Values should be between 0 and 1 and should be level or increasing down the column.
   The first value in the column must be 0.
3. The corresponding water relative permeability (at \( S_w = 1 - S_g \)).
   Values should be between 0 and 1 and should be level or decreasing down the column.
   The last value in the column must be 0.
4. The corresponding gas-water capillary pressure.
   Values should be level or increasing down the column.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB)

Default values (represented by 1*) may be inserted as required in columns 2, 3 and 4. When the table is read in, defaults are replaced by values computed by linear interpolation.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see item 3 of keyword TABDIMS).

The entire table may be defaulted provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

The keyword families

(i) SWOF, SGOF, SLGOF

(ii) SWFN, SGFN, SGWFN, SOF2, SOF3, SOF32D

provide two alternative formats for inputting the saturation functions. Do not mix keywords from the two families in the same run. In gas-water runs, or if you are using the Miscible Flood or Solvent options, the saturation functions must be input with keywords from family (ii).

Note: The critical saturation for each relative permeability function is established by determining the highest saturation in the table for which the relative permeability is notionally zero, that is, less than a small threshold value. Relative permeability values which are lower than this threshold value will be treated as zero. For ECLIPSE 100 and ECLIPSE 300 the default threshold values are \( 1.0 \times 10^{-6} \) and \( 1.0 \times 10^{-20} \) respectively. These values may be modified using the TOLCRIT keyword.
Example

With NTSFUN=2 and NSSFUN ≥9:

<table>
<thead>
<tr>
<th>SGWFN</th>
<th>.0000</th>
<th>.0000</th>
<th>1.0000</th>
<th>.0000</th>
</tr>
</thead>
<tbody>
<tr>
<td>.0400</td>
<td>.0000</td>
<td>.9000</td>
<td>.2000</td>
<td></td>
</tr>
<tr>
<td>.1000</td>
<td>.0022</td>
<td>.8000</td>
<td>.5000</td>
<td></td>
</tr>
<tr>
<td>.2000</td>
<td>.1000</td>
<td>.5000</td>
<td>1.0000</td>
<td></td>
</tr>
<tr>
<td>.3000</td>
<td>.2400</td>
<td>1*</td>
<td>1*</td>
<td></td>
</tr>
<tr>
<td>.5000</td>
<td>.4200</td>
<td>1*</td>
<td>1*</td>
<td></td>
</tr>
<tr>
<td>.6000</td>
<td>.5000</td>
<td>.1125</td>
<td>3.0000</td>
<td></td>
</tr>
<tr>
<td>.7000</td>
<td>.8125</td>
<td>0.0000</td>
<td>3.5000</td>
<td></td>
</tr>
<tr>
<td>.7800</td>
<td>1.0000</td>
<td>0.0000</td>
<td>3.9000</td>
<td></td>
</tr>
<tr>
<td></td>
<td>.00</td>
<td>.0000</td>
<td>1.0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>.12</td>
<td>.0220</td>
<td>.85</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>.22</td>
<td>.1000</td>
<td>.6</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>.32</td>
<td>.2400</td>
<td>.45</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>.52</td>
<td>.4200</td>
<td>.25</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>.62</td>
<td>.5000</td>
<td>.05</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>.72</td>
<td>.8125</td>
<td>0.0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>.80</td>
<td>1.0000</td>
<td>0.0</td>
<td>0</td>
</tr>
</tbody>
</table>

Note: Slow run times may result if the relative permeability changes abruptly over a small saturation interval, especially if IMPES is used. ECLIPSE honors the data input and does not attempt to overcome convergence problems by retabulating over equally spaced saturation intervals.

If chopped relative permeability data is used to minimize dispersion, ECLIPSE performs best in fully-implicit mode if the relative permeability rises steeply from zero (immobility) rather than from a small value.
**SHRATE**

*Activate shear rate form of the polymer shear thinning/thickening logarithmic model*

This keyword is used to activate the shear rate form of the alternative (natural logarithm-based) polymer shear thinning/thickening option and to supply the required shear rate constant. This keyword can only be used in conjunction with keyword PLYSHLOG.

The data comprises NTPVT (see item 2 in RUNSPEC keyword TABDIMS) values that correspond to the shear rate constant in each PVT region, each terminated by a slash (/). The shear rate constants must be positive.

**UNITS**: Dimensionless.

**DEFAULT**: 4.8.

The default value given above is based upon the following expression for the shear rate constant:

\[ C_{\gamma} = C \left( \frac{3n + 1}{4n} \right)^{n-1} \]

This expression is derived using the capillary bundle approach to relating the apparent viscosity of a power-law fluid in porous media to the Darcy velocity. The constant \( C \) can be obtained by comparing viscometer measurements with the apparent viscosities in porous media, and is typically equal to 6.0. Log-log plots of viscosity versus shear rate (or velocity), suggesting that the apparent viscosity is proportional to the Darcy velocity raised to the power \((n-1)\), generate a value of \( n \) that is approximately 0.5. Substituting these values of \( C \) and \( n \) into the expression above provides the default shear rate constant used.

See the "Polymer flood model" in the *ECLIPSE Technical Description* for further information.

**Example**

With NTPVT=1 specified using keyword TABDIMS:

```
SHRATE
4.8 /
```
Dual porosity matrix-fracture coupling

If the dual porosity or dual permeability options are being used (by specifying the DUALPORO or DUALPERM keyword in the RUNSPEC section), this keyword can be used to specify a multiplier to be used in the construction of the matrix-fracture coupling transmissibilities.

If this keyword is used, the sigma factor is applied to the entire grid.

This ‘sigma factor’ may be related to the matrix block size by the expression:

\[ \sigma = 4 \left( \frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2} \right) \]  

where \( l_x \), \( l_y \) and \( l_z \) are typical X, Y and Z dimensions of the matrix blocks.

The relationship was proposed by Kazemi (1976) [Ref. 61].

The dimensions of the matrix blocks used above are not the dimensions of the simulation grid, but of the elements of matrix material in the reservoir.

UNITS: \( \text{m}^2 \) (METRIC), \( \text{ft}^2 \) (FIELD), \( \text{cm}^2 \) (LAB), \( \text{m}^2 \) (PVT-M)

Alternatively, sigma may be treated as a simple history matching factor.

If neither SIGMA nor SIGMAV are specified then the default sigma factor is 0.0 throughout the grid, unless LTOSIGMA is specified in runs using the viscous displacement option.

The dual porosity and dual permeability options are discussed in more detail in "Dual porosity" in the ECLIPSE Technical Description.

Example

```
SIGMA
0.12 /
```

Using Kazemi’s expression, in FIELD units, a value of 0.12 would correspond to 10 ft. matrix blocks.
Matrix-fracture coupling for oil-gas gravity drainage

This keyword can be used to specify an alternative matrix-fracture coupling for matrix cells in which the production mechanism is gravity drainage due to the presence of gas in the fractures.

If this keyword is used, the sigmagd factor is applied to the entire grid.

Different sigmagd values can be specified on a cell by cell basis using the sigmagdv keyword.

The particular formula used to prescribe the alternative is different between ECLIPSE 100 and ECLIPSE 300. In ECLIPSE 100 this keyword is used to provide a different sigma factor for regions swept by gas, while the standard sigma factor is used for regions swept by water. In ECLIPSE 300 however, the keyword is used to provide an alternative sigma factor value when the gravity drainage potential is the dominant pressure term, while allowing smooth variation of sigma as the pressure increases. The next paragraphs provide details on these two formulas.

In ECLIPSE 100 the matrix-fracture transmissibility based on SIGMAGD will be used to calculate the oil flow when the following conditions are satisfied:

- The gravity drainage model is active.
- Oil flow is from the matrix to the fracture.
- The gravity drainage head associated with gas in the fracture is greater than the head associated with water.

Hence regions in which the fractures are swept by gas drain through a transmissibility based on SIGMAGD, and regions swept by water will be produced through a transmissibility based on SIGMA.

The SIGMAGD keyword is only available in three-phase dual porosity runs (keyword DUALPORO in the RUNSPEC section).

This sigma factor may be related to the matrix block size by the expression:

\[ \Sigma = 4 \left( \frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2} \right) \]  

where \( l_x, l_y \) and \( l_z \) are typical X, Y and Z dimensions of the matrix blocks.

The relationship was proposed by Kazemi (1976) [Ref. 61].

The dimensions of the matrix blocks used above are not the dimensions of the simulation grid, but of the elements of matrix material in the reservoir.

Alternatively sigma may be treated as a simple history matching factor.

When calculating the matrix-fracture transmissibility for gravity drainage, the matrix Z-permeabilities are used by default, rather than the X-permeabilities. The pre-99A treatment (using the X-permeabilities) may be restored using item 70 in the OPTIONS keyword.

In ECLIPSE 300, if SIGMA and SIGMAGD are both entered, the program uses an interpolated value, depending on the relative sizes of the capillary pressure and gravity drainage pressure terms. Thus if capillary pressures dominates, SIGMA values are used, whilst if gravity drainage dominates SIGMAGD values are used.

If neither SIGMAGD nor SIGMAGDV are specified, the transmissibilities are based on the sigma value supplied by the SIGMA or SIGMAV keywords, unless LTOSIGMA is specified.
Refer to "Gravity imbibition/drainage" in the ECLIPSE Technical Description for more details.

**Examples**

**Example 1**

```
SIGMA
  0.12 /
```

Using Kazemi's expression, in FIELD units, a value of 0.12 would correspond to 10 ft matrix blocks.

**Example 2**

Smaller SIGMAGD as the gravity drainage is assumed to be dominated by flow only in the z-direction.

```
SIGMAGD
  0.02 /
```
**SIGMAGDV**

**Matrix-fracture coupling for oil-gas gravity drainage**

This keyword can be used to specify an alternative matrix-fracture coupling for matrix cells in which the production mechanism is gravity drainage due to the presence of gas in the fractures.

The keyword should be followed by one positive real number for every grid block in the current box within the top \( \text{NDIVIZ}/2 \) layers of the grid (that is, the matrix cells). Any values input in the bottom half of the grid (the lower \( \text{NDIVIZ}/2 \) layers) will be ignored.

The particular formula used to prescribe the alternative is different between ECLIPSE 100 and ECLIPSE 300. In ECLIPSE 100 this keyword is used to provide a different sigma factor for regions swept by gas, while the standard sigma factor is used for regions swept by water. In ECLIPSE 300 however, the keyword is used to provide an alternative sigma factor value when the gravity drainage potential is the dominant pressure term, while allowing smooth variation of sigma as the pressure increases. The next paragraphs provide details on these two formulas.

In ECLIPSE 100, the matrix-fracture transmissibility based on \( \text{SIGMAGD} \) will be used to calculate the oil flow when the following conditions are satisfied:

- The Gravity Drainage Model is active.
- Oil flow is from the matrix to the fracture.
- The gravity drainage head associated with gas in the fracture is greater than the head associated with water.

Hence regions in which the fractures are swept by gas will drain through a transmissibility based on \( \text{SIGMAGD} \) and regions swept by water will be produced through a transmissibility based on \( \text{SIGMA} \).

The \( \text{SIGMAGDV} \) keyword is only available in three-phase dual porosity runs (keyword \( \text{DUALPORO} \) in the RUNSPEC section).

This sigma factor may be related to the matrix block size by the expression:

\[
\Sigma = 4 \left( \frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2} \right) \quad \text{Eq. 3.174}
\]

where \( l_x \), \( l_y \) and \( l_z \) are typical X, Y and Z dimensions of the matrix blocks.

The relationship was proposed by Kazemi (1976) [Ref. 61].

The dimensions of the matrix blocks used above are not the dimensions of the simulation grid.

**Keywords**

- \( \text{SIGMA} \), \( \text{SIGMAGD} \), \( \text{SIGMAGDV} \)

Alternatively sigma may be treated as a simple history matching factor.

When calculating the matrix-fracture transmissibility for gravity drainage, the matrix Z-permeabilities are used by default, rather than the X-permeabilities. The pre-99A treatment (using the X-permeabilities) may be restored using switch 70 in the OPTIONS keyword.

In ECLIPSE 300, if \( \text{SIGMA} \) and \( \text{SIGMAGD} \) are both entered, the program uses an interpolated value, depending on the relative sizes of the capillary pressure and gravity drainage pressure terms. Thus if capillary pressures dominates, \( \text{SIGMA} \) values are used, whilst if gravity drainage dominates \( \text{SIGMAGD} \) values are used.
If neither $\text{SIGMAGD}$ nor $\text{SIGMAGDV}$ are specified, the transmissibilities will be based on the sigma value supplied by the $\text{SIGMA}$ or $\text{SIGMAV}$ keywords, unless $\text{LTOSIGMA}$ is specified.

When the $\text{SIGMAGD}$ values are output (mnemonic $\text{SIGMAGDV}$ in the $\text{RPTGRID}$ keyword) the first $\text{NDIVIZ}/2$ layers are copied into the lower half of the grid.

Refer to "Gravity imbibition/drainage" in the $\text{ECLIPSE Technical Description}$ for more details.

**Examples**

**Example 1**

```
SIGMAV
 0.12 0.12 0.12 0.12 0.12 0.12 0.12 0.12
 0.06 0.06 0.06 0.06 0.06 0.06 0.06 0.06 /
```

**Example 2**

Set $\text{SIGMAGDV}$ to $\text{SIGMAV}$

```
COPY
  SIGMAV SIGMAGDV 1 8 1 1 1 2 /
```

**Example 3**

Divide the $\text{SIGMAGDV}$ by 4.

```
MULTIPLY
  SIGMAGDV 0.25 1 8 1 1 1 2 /
```
**SIGMATH**

**Dual porosity matrix-fracture conductivity coupling**

This keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section).

If the dual porosity or dual permeability options are being used (keyword DUALPORO or DUALPERM in the RUNSPEC section), this keyword can specify an overall multiplier to be used in the construction of the matrix-fracture thermal conductivity coupling transmissibilities.

The SIGMATH keyword is followed by a value for each matrix cell. Thus \((N_x N_y N_z) / 2\) values are required in all, in normal grid order with the X-index changing fastest.

The dual porosity and dual permeability options are discussed in more detail in "Dual porosity" in the ECLIPSE Technical Description.

The Thermal option is described in "Thermal option" in the ECLIPSE Technical Description.

**Example**

```
SIGMATH
350*0.12 /
```

---

**Keywords**

2062
Dual porosity matrix-fracture coupling

If the dual porosity option is being used (keyword DUALPORO in the RUNSPEC section), this keyword can be used to specify a multiplier to be applied in the construction of the matrix-fracture coupling transmissibilities.

The keyword should be followed by one positive real number for every grid block in the current box within the top NDIVIZ/2 layers of the grid (that is, the matrix cells). Any values input in the bottom half of the grid (the lower NDIVIZ/2 layers) will be ignored.

This sigma factor may be related to the matrix block size by the expression:

\[ \Sigma = 4 \left( \frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2} \right) \]  

Eq. 3.175

where \( l_x, l_y \) and \( l_z \) are typical X, Y and Z dimensions of the matrix blocks.

The relationship was proposed by Kazemi (1976) [Ref. 61].

The dimensions of the matrix blocks used above are not the dimensions of the simulation grid.

UNITS: m^{-2} (METRIC), ft^{-2} (FIELD), cm^{-2} (LAB), m^{-2} (PVT-M)

Alternatively sigma may be treated as a simple history matching factor.

If neither SIGMA nor SIGMAV are specified then the default sigma factor is 0.0 throughout the grid, unless LTOSIGMA is specified in runs using the Viscous Displacement option.

When the SIGMA values are output (mnemonic SIGMAV in the RPTGRID keyword) the first NDIVIZ/2 layers are copied into the lower half of the grid.

When the multi-porosity option is in use (NMATRIX >1) and the grid properties for each porosity are specified separately (NMATOPTS is not used), then each porosity in the grid now has NDIVIZ/(NMATRIX +1) layers. Most of the properties entering the multi-porosity transmissibility or diffusivity between porosities are associated with the corresponding inner matrix layer. However, for SIGMAV this is only true for the matrix-fracture connection as in dual porosity. For multi-porosity connections between matrix cells, the SIGMAV value of the outer matrix is used (see "Matrix-matrix transmissibility and diffusivity" in the ECLIPSE Technical Description). This forces SIGMAV to have NMATRIX contiguous porosities of data, leaving the last porosity unused and preventing the need to fill a porosity with unused default values, as the last porosity can be defaulted by omission.

Example

```
--------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX        1  8  1  1  1  2  /
SIGMAV     0.12 0.12 0.12 0.12 0.12 0.12 0.12
           1.0  1.0  1.0  1.0  1.0  1.0  1.0  /
```

Using Kazemi’s expression, in FIELD units, a value of 0.12 would correspond to 10 ft matrix blocks.
SIMULATE  Turn on simulation

This keyword counteracts a NOSIM keyword in the SCHEDULE section and turns on the timestep simulation.

The keyword has no associated data.

ECLIPSE 100 - Note that this keyword cannot counteract a RUNSPEC section NOSIM keyword or NOSIM in item 3 of the LOAD keyword during a restart run.
**SKIP**

Skip reading of keywords

This keyword causes all subsequent keywords to be ignored, up to the next ENDSKIP keyword.

The skipping of keywords is terminated by the ENDSKIP keyword.

The keywords SKIP, SKIP100 and SKIP300 can be nested provided each instance is matched with an ENDSKIP keyword.

**Example**

```plaintext
-----GRID section
SKIP
CARFIN
'HUGE' 1 20 1 20 1 20 100 100 100 /
ENDFIN
ENDSKIP
```
Skip keywords for ECLIPSE 100

This keyword causes ECLIPSE 100 to ignore all subsequent keywords, up to the next ENDSKP keyword. This keyword is ignored by ECLIPSE 300.

The skipping of keywords is terminated by the ENDSKP keyword.

The keywords SKIP, SKIP100 and SKIP300 can be nested provided each instance is matched with an ENDSKP keyword.

Example

```plaintext
---Props section
SKIP100
INCLUDE
'COMPOSITIONAL.PVO' /
ENDSKIP
```
Skip keywords for ECLIPSE 300

This keyword causes ECLIPSE 300 to ignore all subsequent keywords, up to the next ENDSKP keyword. This keyword is ignored by ECLIPSE 100.

The skipping of keywords is terminated by the ENDSKP keyword.

The keywords SKIP, SKIP100 and SKIP300 can be nested provided each instance is matched with an ENDSKP keyword.

Example

```plaintext
-----Props section
SKIP300
INCLUDE
'BK.PVO' /
ENDSKIP
```
The SKIPREST keyword has no associated data. It instructs ECLIPSE to skip the subsequent keywords in the SCHEDULE section of a restart run until the restart time has been reached. This saves you from having to delete the keywords yourself. It should normally be placed at the very start of the SCHEDULE section - in ECLIPSE 100 parallel runs it is strongly recommended that the keyword is the first one in the SCHEDULE section.

The keyword has no effect in the base run (starting from time zero), but in a restart run it causes ECLIPSE to reset the simulation time to zero and skip the subsequent keywords until a timestepping keyword (for example, TIME, TSTEP, DATES or GASYEAR) advances the simulation time to the restart time. Thereafter the keywords are read in as normal.

Not all keywords are skipped. Some keywords supply data that is not carried over on the restart file, such as:

- VFP tables
  (Keywords VFPSPROD, VFPINJ)

- Reporting instructions
  (Keywords RPTSCHED, RPRTRST or RPTPRINT)

- The “global keywords” that can be entered in any section
  (for ECLIPSE 100 COLUMNS, DEBUG, ECHO, EXTRAPMS, FORMFEED, INCLUDE, MESSAGES, NOECHO, NOWARN and OPTIONS) or
  (for ECLIPSE 300 DEBUG3, ECHO, INCLUDE, MESSAGES, NOECHO and OPTIONS3)

- Annual scheduling files in the Gas Field Operations Model (keywords GASBEGIN, GASEND and any keywords in between)

- The keywords SLAVES, DUMPCUPL and USECUPL in the Reservoir Coupling Facility.

- Well PI Scaling tables (keyword PIMULTAB).

- Scale damage and scale deposition tables (keywords SCDATAB and SCDPTAB).

- The drift flux inclination factor table for multisegment wells (keyword WSEGDPIN).

- Polymer flood model tables (keywords PLYSHEAR, PLYROCKM, PLYDHFLF, PLYMAX, PLYADS and PLYSHLOG).

- The structure of the standard production and injection networks (keywords GRUPNET and GRUPINJE).

- Gas plan tables: keywords GPTABLE, GPTABLEN and GPTABLE3.

- The TUNING keyword.

These keywords must be present in the SCHEDULE section of the restart run, if they are needed in the run. Accordingly, ECLIPSE will read and process these keywords in the usual way, even if they occur between the SKIPREST keyword and the restart time. VFP tables should be entered before the first timestepping keyword (it is best to place them at the start of the SCHEDULE section).

If the SKIPREST keyword is present in a restart run, take care to ensure that it is followed by the timestepping keywords necessary to advance the simulation time from zero to the restart time. There must be a report step corresponding to the restart time, at which ECLIPSE stops skipping keywords and begins...
simulating normally. Thus, if the `DATES` keyword is used to advance the simulation, there **must** be a
`DATES` record with a date identical to the restart date.

**Note:** If `TSTEP`, `GASYEAR` or `GASPERIO` keywords are used to advance the simulation, and some have
been deleted in the restart run, the remaining keywords do not advance the simulation time to reach the
restart time at the correct place in the Schedule data. Advancing the simulation with the `DATES` or `TIME`
keyword is safer in this respect, as these keywords advance the simulation **to** a given time rather than **by** a
given time interval.

**Example**

```
SKIPREST
```
Request the fast flash option

This keyword requests the fast flash option which can be used in EoS compositional runs. See "Fast flash option" in the ECLIPSE Technical Description.

This keyword has no associated data.
SLAVES

Starts the slave reservoir simulation runs

This keyword is used in a reservoir coupling master run to identify the slave reservoir data files, and to indicate where they are located. ECLIPSE starts their simulation processes, which waits for further instructions from the master run. (See “Reservoir coupling” in the ECLIPSE Technical Description.)

The keyword is followed by a data record for each active slave reservoir. Each record contains the following items of data, and should be terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Name of the slave reservoir
   (This should be up to 8 characters long)
   The slave reservoir name is used to identify the slave reservoir in keyword GRUPMAST.

2. The root name of the slave reservoir’s ECLIPSE data file
   (This should be up to 72 characters long)

3. This item should just be ‘*’ in almost all cases; this allows the system to pick which hosts to put the slaves on, depending on the host file supplied. In very specific cases this item should be the host name of the machine on which the slave process is to be run. If a host name has to be specified the following caveats apply:
   (If specified the host name can be up to 32 characters long)
   Enter the master’s host name here, if the slave resides on the same machine as the master run.

4. The path-name of the directory in which the data file is located, from the root directory of the host machine
   (This should be up to 72 characters long)

5. Number of processors the slave is to be run on. If the slave is to be run in parallel this should be an integer corresponding to the first item in the PARALLEL keyword in the slave data set. This value should be set to 1 for runs in serial mode.
   DEFAULT: 1

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Notes

• The SLAVES keyword can only be entered once in a run, so all slave processes must be started together. If at a later date you wish to add another slave reservoir, this may be done in a restart run, adding the new slave reservoir to the others in the SLAVES keyword.

• The SLAVES keyword data is not carried over on the restart file. In restart runs the keyword must be entered again, naming all the slave processes to be activated during the run.

• Up to 20 slave reservoirs are allowed. ECLIPSE must be re-dimensioned if more are required.

• If a slave reservoir is removed in a restart run (by not specifying it again in the SLAVES keyword), any master groups that were originally associated with it will automatically be flagged to read their flows from the Reservoir Coupling file instead. Alternatively, if the slave reservoir is actually no longer producing, you could redefine the master groups as satellite groups with keywords GSATPROD and GSATINJE, giving them zero production and injection rates.
• After reading the SLAVES keyword data, ECLIPSE starts the slave simulation processes. If a particular slave process cannot be started, the master run will print an error message and terminate the other processes. If a slave’s host machine has insufficient free memory to run the process, the operating system may kill the process after MPI has started it. To prevent this situation from hanging the other coupled processes, the master run waits up to 30 seconds for each slave process to send its initial communication. If no response is received within 30 seconds of starting a slave process, the master run prints an error message and terminates the other processes.

• The master run may not include the PARALLEL keyword, so no dataset should contain both the PARALLEL keyword and the SLAVES keyword. Slave simulations may be parallel, but the master must run in serial mode.

Example

<table>
<thead>
<tr>
<th>SLAVES</th>
</tr>
</thead>
<tbody>
<tr>
<td>--</td>
</tr>
<tr>
<td>slave</td>
</tr>
<tr>
<td>datafile</td>
</tr>
<tr>
<td>machine</td>
</tr>
<tr>
<td>directory</td>
</tr>
<tr>
<td>--</td>
</tr>
<tr>
<td>name</td>
</tr>
<tr>
<td>root</td>
</tr>
<tr>
<td>hostname</td>
</tr>
<tr>
<td>of data file</td>
</tr>
<tr>
<td>--</td>
</tr>
<tr>
<td>'upper'</td>
</tr>
<tr>
<td>'base'</td>
</tr>
<tr>
<td>'*'</td>
</tr>
<tr>
<td>'/usr/models/upper'</td>
</tr>
<tr>
<td>'lower'</td>
</tr>
<tr>
<td>'base'</td>
</tr>
<tr>
<td>'*'</td>
</tr>
<tr>
<td>'/usr/models/lower'</td>
</tr>
<tr>
<td>/</td>
</tr>
</tbody>
</table>
SLGOF  

Gas/oil saturation functions versus liquid saturation

The **SLGOF** keyword may be used in runs containing both oil and gas as active phases, to input tables of gas relative permeability, oil-in-gas relative permeability and oil-gas capillary pressure as functions of the liquid saturation. If water is also an active phase in the run, the water/oil saturation functions must be input with keyword **SWOF**.

The data comprises **NTSFUN** (see item 1 in keyword **TABDIMS** in the **RUNSPEC** section) tables of gas/oil saturation functions, each terminated by a slash (/). Each table consists of 4 columns of data:

1. **The liquid saturation.**
   Values should be between 0 and 1 and should increase monotonically down the column. The first value in the column is interpreted as \( S_1 = S_w + S_o \) (\( S_w \) is the connate water saturation, the first saturation value in the **SWOF** keyword, or zero if there is no water in the run). The last value must be 1.0, which corresponds to a gas saturation of zero.

2. **The corresponding gas relative permeability.**
   Values should be between 0 and 1 and should be level or decreasing down the column. The last value in the column must be zero.

3. **The corresponding oil relative permeability when oil, gas and connate water are present.**
   Values should be between 0 and 1 and should be level or increasing down the column. If water is present in the run, the last value in the column \((k_{rog} \text{ at } S_g = 0)\) must be the same as the first value in the \(k_{row}\) column in keyword **SWOF** (that is \(k_{row} \text{ at } S_o = 1 - S_w\)). The first value in the column must be zero.

4. **The corresponding oil-gas capillary pressure.**
   Values should be level or decreasing down the column.

   **UNITS:** bars (METRIC), psi (FIELD), atm (LAB), atm (PVT–M)

Default values (represented by \(1^*\)) may be inserted as required in columns 2, 3 and 4. When the table is read in, defaults are replaced by values computed by linear interpolation.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than **NSSFUN** (see item 3 in keyword **TABDIMS**).

The entire table may be defaulted provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

The keyword families

(i) **SWOF, SGOF** and **SLGOF**

(ii) **SWFN, SGFN, SGWFN, SOF2** and **SOF3**

provide two alternative formats for inputting the saturation functions. Do not mix keywords from the two families in the same run. In gas-water runs, or if you are using the ECLIPSE 100 Miscible Flood or Solvent options, the saturation functions must be input with keywords from family (ii).

For two dimensional tables the following keywords can also be used: **SOF32D, SGF32D** and **SWF32D**.
See "Saturation Functions" in the ECLIPSE Technical Description for more information, and a description of the methods used to compute three phase relative permeabilities.

**Note:** The critical saturation for each relative permeability function is established by determining the highest saturation in the table for which the relative permeability is notionally zero, that is, less than a small threshold value. Relative permeability values which are lower than this threshold value will be treated as zero. For ECLIPSE 100 and ECLIPSE 300 the default threshold values are $1.0 \times 10^{-6}$ and $1.0 \times 10^{-20}$ respectively. These values may be modified using the `TOLCRIT` keyword.

**Example**

With `NTSFUN=2` and `NSSFUN \geq 10`:

```
SLGOF
-- Sl  Krg  Krog  Fcog
 .2200  1.0000  0.00  3.9000
 .3000  .8125  0.00  3.5000
 .4000  .5000  0.00  3.0000
 .5000  .4200  0.00  1*
 .6000  1*  0.00  1*
 .7000  .2400  0.02  1*
 .8000  .1000  0.10  1.0000
 .9000  .0220  0.33  .5000
 .9600  .0000  0.60  .2000
1.0000  .0000  1.00  .0000  / table 1
 .18  1.0000  0.0  0
 .4  .5600  0.0  0
 .5  .4400  0.0  0
 .8  .1100  0.12  0
 .95  .0220  0.62  0
1.00  .0000  1.00  0    / table 2
```
SMF

Specifies cell initial solid composition

This keyword specifies the initial solid composition values for each cell. It can only be used with the \texttt{SOLID} option. If this keyword is used then keyword \texttt{SSOLID} should also be used to specify the amount of initial solids.

The keyword should be followed by $N_x \cdot N_y \cdot N_z \cdot N_c$ values in normal order, with the X index changing fastest. For thermal runs with water, $N_x \cdot N_y \cdot N_z$ extra values are required, all set to zero.

Example

```plaintext
SMF
300*0.00
300*0.00
300*1.00
/```

Keywords

2075
SMRYDIMS Maximum number of summary quantities

The keyword is used to specify the maximum number of summary quantities written to the Summary file (see the SUMMARY section). The data consists of a single item terminated by a slash (/).

NSUMMX The maximum number of vectors written to the Summary file.

DEFAULT: 10000

Example

```
SMRYDIMS
5000 /
```
SMULTX-Z  Alternative transmissibility multipliers for auto-refinement

The SMULTX, SMULTY and SMULTZ keywords specify transmissibility multipliers for auto-refined grids which apply only between cells in the auto-refined grid that lie on an interface between cells in the base grid.

Thus a value of SMULTX for block (IB,JB,KB) in the base grid multiplies the transmissibility between all cells (IA,JA,KA) and (IA+1,JA,KA) in the auto-refined grid which lie on the X+ direction interface between cells (IB,JB,KB) and (IB+1,JB,KB) in the base grid. (That is, auto-refined cell (IA,JA,KA) is contained within the base grid cell (IB,JB,KB), and auto-refined cell (IA+1,JA,KA) is contained within the base grid cell (IB+1,JB,KB).) The behavior of SMULTY and SMULTZ is analogous.

By default, any data for MULTX for example, is inherited from the host cells in the base grid, so that MULTX for cell (IB,JB,KB) applies to all auto-refined cells (IA,JA,KA) contained within the base grid cell (IB,JB,KB).

The SMULT keywords can only be used if item 4 of the AUTOREF keyword in the RUNSPEC section is set to 2. In this case, both MULTX and SMULTX for example can be used together and MULTX will assume the default behavior.

If item 4 of AUTOREF is set to 1, then any MULTX multipliers in the base dataset are treated the same as SMULTX multipliers. The same is true for MULTY and MULTZ multipliers.

The keyword should be followed by one non-negative real number for every grid block in the current input box, and the data must be terminated by a slash (/).

Example

```
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
   5  5  3  8  4  6  /
SMULTX
   18*0.5 /
```

Keywords

---

2077
**SOCRS**

**Scaled critical oil-in-water saturations at miscible conditions**

This keyword allows input of the critical oil-in-water saturation within each cell for the Surfactant Model (see "Surfactant model" in the *ECLIPSE Technical Description*). In the same way that the SOWCR keyword scales the immiscible saturation (SATNUM) tables, the SOCRS keyword scales the SURFNUM tales.

This keyword can only be used when the End Point Scaling option is active (keyword ENDScale in the RUNSPEC section). No directional forms of the keyword are available, and thus if directional scaling is active the SOCRS value will be taken for all directions.

The SOCRS keyword should be followed by one real number for each grid block in the current input box, specifying the critical oil-water saturation within the grid block at miscible conditions. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash(/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

See also keywords SOWCR and SOGCR in the PROPS section.

**Example**

```
-----RUNSPEC section
DIMENS
  1 7 3 /
ENDSCALE
/
-----PROPS section
--------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3
SOCRS
10*0.05 /
```
Oil saturation functions (two-phase)

The SOF2 table has two uses:

- to specify the relative permeability of oil in two-phase models (oil-gas or oil-water)
- to specify the relative permeability of miscible hydrocarbon in solvent models.

SOF2 should not be used to specify the oil’s relative permeability in three-phase systems.

The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) tables of oil saturation functions, each terminated by a slash (/).

Each table consists of 2 columns of data:

1. The saturation \( S_p \).
   - Values should be between 0 and 1 and should increase monotonically down the column.

2. The corresponding relative permeability \( k_{rp} \).
   - Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0.

When the keyword is used in two-phase models, column 1 is the saturation of the oil phase \( S_p = S_o \) and column 2 is the relative permeability of the oil phase \( k_{rp} = k_{ro} \). When the keyword is used in solvent models, column 1 is the total hydrocarbon saturation (including solvent) \( S_p = S_n = S_o + S_g + S_s \) and column 2 is the relative permeability of the miscible hydrocarbon with respect to water \( k_{rp} = k_{rn} \).

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see item 3 of keyword TABDIMS).

The entire table may be defaulted provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

The keyword families

(i) SWOF, SGOF, SLGOF

(ii) SWFN, SGFN, SGWFN, SOF2, SOF3, SOF32D

provide two alternative formats for inputting the saturation functions. Do not mix keywords from the two families in the same run. SOF2 must be input with keywords from family (ii).

For more information, refer to "Saturation functions" and "Solvent model" in the ECLIPSE Technical Description.

Note: The critical saturation for each relative permeability function is established by determining the highest saturation in the table for which the relative permeability is notionally zero, that is, less than a small threshold value. Relative permeability values which are lower than this threshold value will be treated as zero. For ECLIPSE 100 and ECLIPSE 300 the default threshold values are \( 1.0 \times 10^{-6} \) and \( 1.0 \times 10^{-20} \) respectively. These values may be modified using the TOLCRIT keyword.

Example

With NTSFUN=2 and NSSFUN \( \geq 12 \):

Keywords
<table>
<thead>
<tr>
<th>Value</th>
<th>0.0000</th>
<th>0.0000</th>
<th>0.0043</th>
<th>0.0048</th>
<th>0.0529</th>
<th>0.0649</th>
<th>0.1130</th>
<th>0.1250</th>
<th>0.3450</th>
<th>0.4000</th>
<th>0.7000</th>
<th>0.7000</th>
<th>1.0000</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0</td>
<td>0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.2</td>
<td>0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.38</td>
<td>0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.4</td>
<td>0.004</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.48</td>
<td>0.02</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.5</td>
<td>0.036</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.58</td>
<td>0.1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.6</td>
<td>0.146</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.68</td>
<td>0.33</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.7</td>
<td>0.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.74</td>
<td>0.60</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.78</td>
<td>1.00</td>
<td>/</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.8</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.2000</td>
<td>0.0000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.3800</td>
<td>0.0043</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.4000</td>
<td>0.0048</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.4800</td>
<td>0.0529</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.5000</td>
<td>0.0649</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.5800</td>
<td>0.1130</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.6000</td>
<td>0.1250</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.6800</td>
<td>0.3450</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.7000</td>
<td>0.4000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.7400</td>
<td>0.7000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.7800</td>
<td>1.0000</td>
<td>/</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**SOF3**

**Oil saturation functions (three-phase)**

The data comprises NTSFUN (see item 1 of keyword **TABDIMS** in the RUNSPEC section) tables of oil saturation functions, each terminated by a slash (/).

Each table consists of 3 columns of data:

1. The oil saturation.
   
   Values should be between 0 and 1 and should increase monotonically down the column. The maximum oil saturation should be equal to 1 - S\textsubscript{wco}, where S\textsubscript{wco} is the connate water saturation.

2. The corresponding oil relative permeability for regions where only oil and water are present.
   
   Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0.

3. The corresponding oil relative permeability for regions where only oil, gas and connate water are present.
   
   Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0.

Default values (represented by 1*) may be inserted as required in columns 2 and 3. When the table is read in, defaults are replaced by values computed by linear interpolation. Thus, in the table

\[
\begin{array}{ccc}
\text{SOF3} & 0 & 0 \\
& 0.2 & 1* \\
& 1 & 0.6 & /
\end{array}
\]

the default value (the second entry in column 3) is converted to 0.12.

The entire table may be defaulted provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see item 3 of keyword **TABDIMS**).

The maximum values in columns 2 and 3 both represent the oil relative permeability at maximum oil saturation (S\textsubscript{o} = 1 - S\textsubscript{wco}) and should therefore have the same value.

The keyword families

(i) **SWOF, SGOF, SLGOF**

(ii) **SWFN, SGFN, SGWFN, SOF2, SOF3, SOF32D**

   **SWF3, SGF3, SOF3** (**IKU3P** and **ODD3P** options)

provide two alternative formats for inputting the saturation functions. Do not mix keywords from the two families in the same run. In gas-water runs, or if you are using the ECLIPSE 100 miscible flood or solvent options, the saturation functions must be input with keywords from family (ii).

See "Saturation functions" in the **ECLIPSE Technical Description** for a description of the method used to compute the oil relative permeability in blocks where all three phases are present.

See also keywords **BAKER1, BAKER2, STONE1, STONE2, IKU3P** and **ODD3P**.
Note that SOF3 should be used only in three phase cases. In oil/gas and oil/water cases use SOF2.

The keyword SOF3D may be used in place of SOF3 to specify the three-phase oil relative permeabilities directly as a function of water and gas saturations in a two dimensional table.

**ECLIPSE 300**

If the IKU3P or the ODD3P option is selected, column 3 should correspond to the oil relative permeability in the presence of gas only; that is, in the absence of connate water.

**Note:** The critical saturation for each relative permeability function is established by determining the highest saturation in the table for which the relative permeability is notionally zero, that is, less than a small threshold value. Relative permeability values which are lower than this threshold value will be treated as zero. For ECLIPSE 100 and ECLIPSE 300 the default threshold values are $1.0 \times 10^{-6}$ and $1.0 \times 10^{-20}$ respectively. These values may be modified using the TOLCRIT keyword.

### Example

With NTSFUN=1 and NSSFUN $\geq$12:

```
SOF3
  .0000  .0000  .0000
  .2000  .0000  .0000
  .3800  1*     .0000
  .4000  .0048  1*
  .4800  1*     .0200
  .5000  .0649  1*
  .5800  1*     .1000
  .6000  .1250  1*
  .6800  1*     .3300
  .7000  .4000  1*
  .7400  1*     .6000
  .7800  1.0000  1.0000 /
```
SOF32D  2D table of oil relative permeability

This keyword is an alternative to the SOF3 keyword, allowing the three-phase relative permeability of oil to be input directly as a function of water and gas saturations in a two dimensional table.

The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) tables of oil relative permeability, tabulated against both water and gas saturations, each terminated by a slash (/).

Each table consists of a number of records:

Record 1  The water saturation values

Values should be between 0 and 1 and should increase monotonically. The number of water saturation values should not exceed NSSFUN (see item 3 of keyword TABDIMS). The minimum water saturation should be set to \( S_{wco} \), the connate water saturation.

Subsequent records (a maximum of NSSFUN records):

The gas saturation.

Followed by the oil relative permeability values for this gas saturation at all the corresponding water saturations entered in Record 1.

The oil relative permeabilities should be monotonically decreasing, with a final value of zero. The number of oil relative permeability values should not exceed the number of water saturation values entered in Record 1. The record may be truncated at any point provided that the last relative permeability value is zero.

The columns of data must also comply with the following constraints:

• The gas saturation (the first item in the second and subsequent records) should increase monotonically down the column. The oil relative permeability for each water saturation should decrease monotonically down the column and should end in a zero value.

• The relative permeability must also be zero for any point on the table in which \( S_{wat} + S_{gas} \geq 1.0 \).

• Default values may not be placed within the table.

The entire table may be defaulted provided that the table is not the first. Defaulted tables are replaced with a copy of the previous table.

Since the three-phase oil relative permeabilities are specified directly by this keyword, any keywords controlling the three-phase relative permeability model (STONE1, STONE2) will be ignored.

The keyword cannot at present be used with the end-point scaling, hysteresis and miscible options.

Example

With NTSFUN=1 and \( \geq NSSFUN \geq 11 \):

```
SOF32D

-----------------------------------SWAT-----------------------------------
0.22  0.27  0.32  0.37  0.42  0.47  0.52  0.57  0.72  0.77  0.78 /
--SGAS
0.00  1.000  0.625  0.345  0.207  0.113  0.083  0.053  0.023  0.002  0.001  0.000 /
0.05  0.555  0.337  0.210  0.110  0.078  0.047  0.021  0.004  0.001  0.000 /
0.10  0.330  0.212  0.106  0.074  0.042  0.019  0.003  0.002  0.000 /
0.15  0.215  0.103  0.069  0.036  0.017  0.003  0.002  0.001  0.000 /
0.20  0.100  0.065  0.031  0.015  0.002  0.001  0.000 /
```

Keywords

2083
<table>
<thead>
<tr>
<th>Value</th>
<th>Probability</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.25</td>
<td>0.060 0.025 0.014 0.002 0.001 0.001 0.000 /</td>
</tr>
<tr>
<td>0.30</td>
<td>0.020 0.012 0.001 0.001 0.001 0.000 /</td>
</tr>
<tr>
<td>0.35</td>
<td>0.010 0.001 0.001 0.001 0.000 /</td>
</tr>
<tr>
<td>0.40</td>
<td>0.000 0.000 0.000 0.000 /</td>
</tr>
</tbody>
</table>
Scaled critical oil-in-gas saturations

The SOGCR and ISOGCR keywords specify the critical oil-in-gas saturation (that is the largest oil saturation for which the oil relative permeability is zero in an oil-gas-connate water system) within each grid cell. The oil flow across each grid face in an oil-gas-connate water system is calculated from a transformed oil-gas relative permeability curve obtained by linearly scaling the tabulated oil-gas relative permeability data between the new critical oil-in-gas saturation defined using an SOGCR keyword, and the new maximum oil saturation (see under the keywords SWL, SGL in the PROPS section). The SOGCR keyword also permits scaling of the oil-gas relative permeability table used in computing the flow of oil between grid blocks and well connections.

Seven keywords are provided to allow the critical oil-in-gas saturation for each grid cell to be scaled independently.

These keywords are:

- **SOGCR**: Critical oil-gas saturation for the non-directional saturation function tables
- **SOGCRX**: Critical oil-gas saturation for the +X face of each grid cell
- **SOGCRX-**: Critical oil-gas saturation for the -X face of each grid cell
- **SOGCRY**: Critical oil-gas saturation for the +Y face of each grid cell
- **SOGCRY-**: Critical oil-gas saturation for the -Y face of each grid cell
- **SOGCRZ**: Critical oil-gas saturation for the +Z face of each grid cell
- **SOGCRZ-**: Critical oil-gas saturation for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the critical oil-in-gas saturations for flows in the +R,-R,+T,-T,+Z,-Z directions respectively.

None of the SOGCR keywords should be used if either oil or gas is absent from the model. The keyword ENDSCALE must be specified in RUNSPEC.

If both directional and irreversible end-point scaling is requested, using switches 'DIRECT' and 'IRREVERS' in the ENDSCALE keyword in RUNSPEC, then the critical oil-gas saturations for oil flow across the +X and -X faces of a grid cell are set using the keywords SOGCRX, SOGCRX-. Similarly SOGCRY, SOGCRY- are used for the +Y, -Y faces and the keywords SOGCRZ, SOGCRZ- for the +Z, -Z faces. Scaling of the well connection relative permeability tables for any well connections in the grid cell are performed using the SOGCR keyword.

When the end-point scaling is requested to be directional and reversible, using the switch 'DIRECT' but not 'IRREVERS' in ENDSCALE, the keyword SOGCRX defines the critical oil-gas saturations for the +X and -X faces of each grid cell. Keyword SOGCRY is used for both the +Y,-Y faces and the keyword SOGCRZ, for both the +Z,-Z faces of each grid cell. Scaling of well connection relative permeability tables uses data entered under the SOGCR keyword.

If ENDSCALE is specified in RUNSPEC without the switch 'DIRECT', then none of the directional forms of the SOGCR keyword are allowed. The SOGCR keyword then defines the new critical oil-gas saturation for oil flow out of each face of the grid cell and for oil flows between the grid cell and well connections.
Each **SOGCR** keyword should be followed by one real number for each grid block in the current input box specifying the critical oil-gas saturation within the grid block. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash(/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If any of the keywords **SOGCR**, **SOGCRX**, **SOGCRY**, **SOGCRY**, **SOGCRZ**, **SOGCRZ** are omitted in a model where they could be accepted then the critical oil-in-gas saturation for an oil relative permeability calculation defaults to the value used in the appropriate relative permeability table.

**ECLIPSE 100**

If the hysteresis model (switch 'HYSTER' in keyword **SATOPTS**) is active then the **SOGCR** data scales the drainage curve. The imbibition curve can be scaled by using the equivalent **ISOGCR** (**ISOGCRX**, **ISOGCRY**, ...) keyword.

**ECLIPSE 300**

If the **ODD3P** relative permeability and capillary pressure hysteresis model has been selected, the primary end-point scaling is specified using the **PSORG**, **PSGRO**, **PSORW**, **PSWRO**, **PSGRW** and **PSWRG** keywords and the hysteresis end-point scaling is specified using the **HSORG**, **HSGRO**, **HSORW**, **HSWRO**, **HSGRW** and **HSWRG** keywords.

See also keywords **SGL**, **SWL**, **ENPTVD** in the **PROPS** section.

See "Saturation table scaling" in the **ECLIPSE Technical Description** for further details.

**Note:** When consistency warning messages involving **SOGCR** are reported by ECLIPSE, in some cases this may be referred to using the alternative name **SOCRG**. (It should be noted that this is not accepted as an alternative keyword name.)

---

**Example**

```plaintext
-----RUNSPEC section
DIMENS
1 7 3 /
ENDSCALE
/
-----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
1 1 2 7 1 3
SOGCR
5*0.24 10*0.21 3*0.20 /
```

---

**Keywords**

---

**2086**
SOIL

Specifies cell initial oil saturation values

The keyword should be followed by one real number for each grid block in the current input box, terminated with a slash (/).

This keyword specifies the initial oil saturation values for each cell.

This keyword may be used to specify the initial solution explicitly, as an alternative to equilibration using EQUIL. It is expected if the PRESSURE keyword is present, indicating explicit initialization, and if an oil phase is present. The same information may be obtained from a Restart file using the GETSOL keyword.

A sufficient set of the SOIL, SWAT and SGAS keywords should be entered. For example, for a three phase run any pair may be used; for a two-phase oil/gas run use either SOIL or SGAS.

For the mixed hydrostatic equilibration option, initial saturations should be specified together with items 1 and 2 of the EQUIL keyword and the NEI keyword. For more information, refer to "Initializing the study" in the ECLIPSE Technical Description.

When using explicit initialization, you must ensure that the initial solution is stable and physically reasonable.

Example

```
SOIL
4000*0.17 /
```

Keywords

ECLIPSE Reference Manual

2087
SOILR

Specifies cell initial residual oil saturation values

This keyword specifies the initial residual oil saturation values for each cell. It can only be used with the SOR option. This keyword is optional, but if used, a value should be given for every cell in the grid.

The keyword should be followed by one real number for each grid block in the current input box, terminated with a slash (/).

Example

```
SOR
4000*0.01 /
```
Run contains solids

This keyword indicates that the run contains a solid phase. This keyword has no associated data.

See "Solid phase" in the ECLIPSE Technical Description.
**SOLIDADS**

**Solid adsorption function**

The data comprises `NTSFUN` (see keyword `TABDIMS`) tables of solid adsorption functions describing the adsorption of solid by the rock formation. See "Solid phase" in the *ECLIPSE Technical Description* for further information.

Each table consists of 2 columns of data and is terminated by a slash (/):

1. The local solid concentration in the solution surrounding the rock.
   - The values should increase monotonically down the column.
   - **UNITS:** kg/m\(^3\) (METRIC), lb/ft\(^3\) (FIELD), gm/cc (LAB), kg/m\(^3\) (PVT-M)

2. The corresponding concentration of solid adsorbed by the rock formation.
   - These values should not decrease down the column.
   - **UNITS:** kg/kg (METRIC), lb/lb (FIELD), gm/gm (LAB), kg/kg (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than `NSSFUN` (see keyword `TABDIMS`).

The adsorbed concentration is measured as weight of solids per weight of rock. The rock density can be set with the `ROCKDEN` keyword.

If this keyword is not used then it is assumed that all solids are adsorbed to the rock. If this keyword is used, but the rock density is set to zero (or not set) then it is assumed that no solids are adsorbed to the rock.

**Example**

With `NTSFUN=2` and `NSSFUN` $\geq 6$

```
SOLIDADS
.0000 .00000
.0000 .00001
.0003 .00005
.0005 .00007
.0008 .00008
.0010 .00008 /

.0000 .00000
.0010 .00010 /
```

---

**Keywords**

2090
SOLIDMMC  Mobility multiplier as a function of adsorbed solid concentration

The data comprises NTSFUN (see keyword TADBIMS) tables of solid mobility factors describing the reduction in mobility caused by solid adsorbed on to the rock formation. See “Solid phase” in the ECLIPSE Technical Description for further information.

Each table consists of 2 columns of data and is terminated by a slash (/):

1. The concentration of solid adsorbed by the rock formation.
   - These values should increase monotonically down the column.
   - UNITS: kg/kg (METRIC), lb/lb (FIELD), gm/gm (LAB), kg/kg (PVT-M)

2. The corresponding mobility multiplier.
   - There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see keyword TADBIMS).

The adsorbed concentration is measured as weight of solids per weight of rock. The rock density must be set with ROCKDEN when this keyword is used.

The amount of solids adsorbed can be defined by the SOLIDADS keyword.

SOLIDMMS can be used as an alternative to this keyword; in that keyword the mobility multiplier is specified as a function of the adsorbed solid saturation.

Example

With NTSFUN=2 and NSSFUN ≥ 6

```
SOLIDMMC  0.0000  1.00
          0.0001  0.99
          0.0003  0.81
          0.0005  0.75
          0.0008  0.36
          0.0010  0.10 /
          0.0000  1.00
          0.0010  0.10 /
```
**SOLIDMMS**

**Mobility multiplier as a function of adsorbed solid saturation**

The data comprises NTSFUN (see keyword `TABDIMS`) tables of fluid mobility factors, describing the reduction in mobility caused by solid adsorbed on to the rock formation. See "Solid phase" in the *ECLIPSE Technical Description* for further information.

Each table consists of 2 columns of data and is terminated by a slash (/):

1. The saturation of solid adsorbed by the rock formation.
   - These values should increase monotonically down the column.
   - UNITS: kg/kg (METRIC), lb/lb (FIELD), gm/gm (LAB), kg/kg (PVT-M)

2. The corresponding mobility multiplier.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see keyword `TABDIMS`).

The amount of solids adsorbed can be defined by the `SOLIDADS` keyword.

`SOLIDMMS` can be used as an alternative to this keyword; in that case the mobility multiplier is specified as a function of the adsorbed solid concentration.

**Example**

With NTSFUN=2 and NSSFUN ≥ 6

```
SOLIDMMS
0.0  1.00
0.1  0.99
0.3  0.81
0.5  0.75
0.8  0.36
1.0  0.00 /

0.0  1.0
1.0  0.0 /
```

*Keywords*

2092
**SOLUAQA**

**Specify properties of water multi-component gas system**

This keyword enables the default aqueous phase property data to be overwritten for the **GASSOL** option. The data for each soluble component consists of two records. The first record is the component index. It is followed by a table consisting of 2 columns of data. Each record is terminated by a slash. A null record, that is a record with no data before the terminating slash, must end this keyword.

Note that the solubility data may be defaulted for any given soluble component. In such a case, a default set of pressures and corresponding solubilities is generated. If the solubilities for all soluble components are defaulted, then each table comprises twenty rows of data. If the solubilities for some (but not all) soluble components are defaulted, then the number of rows in the defaulted solubility table(s) equals the maximum number of rows from the remaining input solubility tables. Since the default solubilities depend strongly on the data entered in the **PROPS** section, this may lead to unrealistic values if the component data is not accurate enough. Henceforth, you are strongly encouraged to enter your own values for the solubility data. Some recommended data, for CO$_2$, is included at the end of this section. For further details concerning the calculation of default solubilities, see "GASSOL solution in the aqueous phase" in the **ECLIPSE Technical Description**.

**Record 1**

One item of data:

1. The component index defined by the **CNames** keyword.

**Record 2**

A table consisting of 2 columns of data.

1. Pressure.

   Values should increase monotonically down the column.

   **UNITS**: barsa (METRIC), psia (FIELD), atm (LAB), atma (PVT-M)

2. The dissolved gas solution ratio at the specified pressure.

   This is expressed as the volume of gas component at surface conditions per volume of water at surface conditions for a saturated solution.

   **UNITS**: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)

See also "GASSOL solution in the aqueous phase" in the **ECLIPSE Technical Description**.

This table may be omitted if the solubility data is to be defaulted.

**Example**

---Metric units

```plaintext
SOLUAQA
2/
  1.0132  0.3038
  533.7694  36.5522
  1013.2500  43.4860
/
4/
  1.0132  0.2947
  533.7694  35.4555
```

**Keywords**

2093
null record terminates table

---Some recommended data for CO2
--Pressure vs Solubility
--METRIC UNITS
1.0132 0.7046
54.2889 22.1123
107.5645 29.1687
160.8401 31.9124
214.1157 33.2806
267.3913 34.2442
320.6670 35.1419
373.9426 36.0393
427.2182 36.9367
480.4938 37.8341
533.7694 38.7314
587.0451 39.6288
640.3207 40.5262
693.5963 41.4236
746.8719 42.3209
800.1475 43.2183
853.4231 44.1157
906.6988 45.0131
959.9744 45.9105
1013.2500 46.8078

Recommended CO₂ solubility data (metric units)
Specify properties of water-gas system

This keyword enables the default aqueous phase property data to be overwritten. The data consists of a single solubility table composed of a series of records terminated with a slash (/).

The dissolved gas can either be CO₂ (CO2SOL) or H₂S (H2SSOL).

The first line of the table should represent water at standard conditions when dissolved gas is at its minimum. The simulator does not assume that the minimum gas in water is zero; if you want to specify that the amount of gas in water is zero at standard conditions, you should set item 2 to zero in the first line of the table.

The contents of each column are as follows:

1. Pressure.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
2. The dissolved gas solution ratio at the specified pressure.
   This is expressed as the volume of gas at surface conditions per volume of water at surface conditions.
   UNITS: sm³/sm³ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm³/sm³ (PVT-M)
3. The formation volume of water saturated with gas at the specified pressure.
   UNITS: rm³/sm³ (METRIC), rb/stb (FIELD), rcc/scc (LAB), rm³/sm³ (PVT-M)
4. The viscosity of water saturated with gas at the specified pressure.
   UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)
5. The compressibility of water saturated with gas at the specified pressure.
   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

Note: Default values are only generated for CO₂.

Note: Default values (represented by 1*) may be inserted as required in columns 2 to 5. If all values in a given column are defaulted, they are replaced by values calculated using the rules used when the keyword is omitted as stated below. Otherwise, they are replaced by values computed by linear interpolation.

If this keyword is omitted, a default table is generated. The values for item 2 are calculated from the correlations given by Chang, Coats and Nolen [Ref. 18]. For the values of item 3, we first compute the density of pure water [Ref. 62], then we use Ezrokhi’s method to calculate the effect of salt and CO₂, [Ref. 133]. When setting item 115 of OPTIONS3, the values for item 3 are calculated using the correlation given by [Ref. 18]. The value for items 4 and 5 are taken from the PVTW data.

The salinity values for the correlations used in items 2 and 3 can be input using either the SALINITY or SALINITR keyword, defaulting to zero.

See also "CO₂/H₂S solution in the aqueous phase" in the ECLIPSE Technical Description.
### Example

<table>
<thead>
<tr>
<th>SOLUBILI</th>
<th>Pres</th>
<th>Rs</th>
<th>FVF</th>
<th>Visc</th>
<th>Compress</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>[bars]</td>
<td>[sm3/sm3]</td>
<td>[rm3/sm3]</td>
<td>[cP]</td>
<td>[1/bars]</td>
</tr>
<tr>
<td>1.013</td>
<td>0.7537</td>
<td>1.0057</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>54.29</td>
<td>23.09</td>
<td>1.0037</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>107.6</td>
<td>30.05</td>
<td>1.0015</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>160.8</td>
<td>32.65</td>
<td>.99923</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>214.1</td>
<td>33.94</td>
<td>.99701</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>267.4</td>
<td>34.88</td>
<td>.99482</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>320.7</td>
<td>35.78</td>
<td>.99266</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>373.9</td>
<td>36.68</td>
<td>.99052</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>427.2</td>
<td>37.59</td>
<td>.98842</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>480.5</td>
<td>38.49</td>
<td>.98635</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>533.8</td>
<td>39.39</td>
<td>.98431</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>587.0</td>
<td>40.29</td>
<td>.98230</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>640.3</td>
<td>41.20</td>
<td>.98032</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>693.6</td>
<td>42.10</td>
<td>.97837</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>746.9</td>
<td>43.00</td>
<td>.97644</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>800.1</td>
<td>43.91</td>
<td>.97455</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>853.4</td>
<td>44.81</td>
<td>.97268</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>906.7</td>
<td>45.71</td>
<td>.97084</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>960.0</td>
<td>46.61</td>
<td>.96903</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>1013.</td>
<td>47.52</td>
<td>.96725</td>
<td>.30</td>
<td>.4E-4</td>
<td></td>
</tr>
<tr>
<td>/</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
SOLUBILS  Specify properties of water-CO2 system with salt dependencies

This keyword enables the default aqueous phase property data to be overwritten. The data consists of a solubility table composed of a series of 2 records, each terminated by a slash. A null record, that is a record with no data before the terminating slash, must end this keyword.

The first line of record 2 should represent water at standard conditions when dissolved CO\textsubscript{2} is at its minimum for a given salinity. The simulator does not assume that the minimum CO\textsubscript{2} in water is zero; if you want to specify that the amount of CO\textsubscript{2} in water is zero at standard conditions then you should set item 2 to zero in each record 2 of the table.

If the brine option is active (keyword BRINE), the salinity values used to interpolate in this table are taken from the brine tracer. Otherwise, they are assumed to be zero.

If the brine option is active, the keywords PVTWSALT and BDENSITY are not allowed as the keyword SOLUBILS performs their function.

Record 1

Three items of data:

1. The salt concentration.
   Values should increase monotonically down the column.
   UNITS: kg/sm\textsuperscript{3} (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm\textsuperscript{3} (PVT-M)

2. Brine surface density.
   UNITS: kg/m\textsuperscript{3} (METRIC), lb/ft\textsuperscript{3} (FIELD), gm/cc (LAB), kg/m\textsuperscript{3} (PVT-M)
   DEFAULT: the brine takes the surface density of water, as specified in the DENSITY (or GRAVITY) keyword.

3. The reference salt concentration for stock tank water, c_{s ref}.
   This item is only read in the first record of the table.
   UNITS: kg/sm\textsuperscript{3} (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm\textsuperscript{3} (PVT-M)
   DEFAULT: Salt concentration specified in item 1 of the first record of the table (typically zero).

Record 2

A table consisting of 5 columns of data.

1. Pressure
   Values should increase monotonically down the column.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2. The dissolved CO\textsubscript{2} solution ratio at the specified pressure and salt concentration.
   This is expressed as the volume of CO\textsubscript{2} at surface conditions per volume of water at surface conditions for a salt-saturated solution.
3. The formation volume factor of water saturated with CO\textsubscript{2} at the specified pressure and salt concentration.

UNITS: m³/sm³ (METRIC), Mcf/stb (FIELD), scc/scc (LAB), m³/sm³ (PVT–M)

4. The viscosity of water saturated with CO\textsubscript{2} at the specified pressure and salt concentration.

UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT–M)

5. The compressibility of water saturated with CO\textsubscript{2} at the specified pressure and salt concentration.

UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT–M)

See also "CO\textsubscript{2}/H\textsubscript{2}S solution in the aqueous phase" in the ECLIPSE Technical Description.

Example

```
---Metric units
SOLUBILS
  0.0 999.0 / 0.0 1.0132 0.3038 1.03504 0.30 0.4E-4
  533.7694 36.5522 0.98431 0.30 0.4E-4
  1013.2500 43.4860 0.96725 0.30 0.4E-4

  10. 1020.0 / 1.0132 0.2947 1.03504 0.30 0.4E-4
  533.7694 35.4555 0.98431 0.30 0.4E-4
  1013.2500 42.1814 0.96725 0.30 0.4E-4

  20. 1030.0 / 1.0132 0.2879 1.03504 0.30 0.4E-4
  533.7694 34.6367 0.98431 0.30 0.4E-4
  1013.2500 41.2073 0.96725 0.30 0.4E-4

/ null record terminates table
```
SOLUBILT Specify properties of water-CO2 system with temperature dependencies

This keyword enables the default aqueous phase property data to be overwritten. The data consists of a solubility table composed of a series of 2 records, each terminated by a slash. A null record, that is a record with no data before the terminating slash, must end this keyword.

The first line of record 2 should represent water at standard conditions when dissolved CO$_2$ is at its minimum for a given temperature. The simulator does not assume that the minimum CO$_2$ in water is zero; if you want to specify that the amount of CO$_2$ in water is zero at standard conditions then you should set item 2 to zero in each record 2 of the table.

If the TEMPVD keyword is used to specify a variable temperature profile with depth, this is used as temperature values to interpolate in this table. Otherwise, the values specified in RTEMP are used.

The salinity values for the correlations used in items 2 and 3 of record 2 can be input using either the SALINITY or SALINITR keyword, defaulting to zero.

**Record 1**

One item of data:

1. The temperature
   - Values should increase monotonically down the column.
   - UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

**Record 2**

A table consisting of 5 columns of data.

1. Pressure
   - Values should increase monotonically down the column.
   - UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
2. The dissolved CO$_2$ solution ratio at the specified pressure and temperature.
   - This is expressed as the volume of CO$_2$ at surface conditions per volume of water at surface conditions.
   - UNITS: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)
3. The formation volume factor of water saturated with CO$_2$ at the specified pressure and temperature.
   - UNITS: rm$^3$/sm$^3$ (METRIC), rb/stb (FIELD), rcc/scc (LAB), rm$^3$/sm$^3$ (PVT-M)
4. The viscosity of water saturated with CO$_2$ at the specified pressure and temperature.
   - UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)
5. The compressibility of water saturated with CO$_2$ at the specified pressure and temperature.
   - UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

See also "CO$_2$/H$_2$S solution in the aqueous phase" in the ECLIPSE Technical Description.
### Example

```plaintext
--- Metric units
SOLUBILT
  122. /
    1.0132  0.3038  1.03504  0.30  0.4E-4
  533.7694  36.5522  0.98431  0.30  0.4E-4
  1013.2500  43.4860  0.96725  0.30  0.4E-4
  257. /
    1.0132  0.2947  1.03504  0.30  0.4E-4
  533.7694  35.4555  0.98431  0.30  0.4E-4
  1013.2500  42.1814  0.96725  0.30  0.4E-4
  392. /
    1.0132  0.2879  1.03504  0.30  0.4E-4
  533.7694  34.6367  0.98431  0.30  0.4E-4
  1013.2500  41.2073  0.96725  0.30  0.4E-4
/ null record terminates table
```
**SOLUCO2H  Specify properties of water-CO2 system with H2S dependencies**

This keyword enables the default aqueous phase property data to be overwritten. The data consists of a solubility table composed of a series of 2 records, each terminated by a slash. A null record, that is a record with no data before the terminating slash, must end this keyword.

The first line of record 2 should represent water at standard conditions when dissolved CO\textsubscript{2} is at its minimum for a given H\textsubscript{2}S aqueous mole fraction. The simulator does not assume that the minimum CO\textsubscript{2} in water is zero; if you want to specify that the amount of CO\textsubscript{2} in water is zero at standard conditions then you should set item 2 to zero in each record 2 of the table.

**Record 1**

One item of data:

1. The H\textsubscript{2}S mole fraction.

**Record 2**

A table consisting of 5 columns of data.

1. Pressure.
   
   Values should increase monotonically down the column.
   
   UNITS: barsa (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)

2. The dissolved CO\textsubscript{2} solution ratio at the specified pressure and H\textsubscript{2}S concentration.
   
   This is expressed as the volume of CO\textsubscript{2} at surface conditions per volume of water at surface conditions for a salt-saturated solution.
   
   UNITS: sm\textsuperscript{3}/sm\textsuperscript{3} (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm\textsuperscript{3}/sm\textsuperscript{3} (PVT-M)

3. The formation volume factor of water saturated with CO\textsubscript{2} at the specified pressure and H\textsubscript{2}S concentration.
   
   UNITS: rm\textsuperscript{3}/sm\textsuperscript{3} (METRIC), rb/stb (FIELD), rcc/scc (LAB), rm\textsuperscript{3}/sm\textsuperscript{3} (PVT-M)

4. The viscosity of water saturated with CO\textsubscript{2} at the specified pressure and H\textsubscript{2}S concentration.
   
   Note that Ezrokhi’s method may be used to generate the default water viscosity using item 164 of OPTIONS3. A default value of 0.3 cP will be used in the absence of this option switch.
   
   UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

5. The compressibility of water saturated with CO\textsubscript{2} at the specified pressure and H\textsubscript{2}S concentration.
   
   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

See also "CO\textsubscript{2}/H\textsubscript{2}S solution in the aqueous phase" in the *ECLIPSE Technical Description*.

**Example**

---

---Metric units
SOLUCO2H

---

**Keywords**

ECLIPSE Reference Manual
<p>| | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0/</td>
<td>1.0132</td>
<td>0.3038</td>
<td>1.03504</td>
<td>0.30</td>
<td>0.4E-4</td>
</tr>
<tr>
<td></td>
<td>533.7694</td>
<td>36.5522</td>
<td>0.98431</td>
<td>0.30</td>
<td>0.4E-4</td>
</tr>
<tr>
<td></td>
<td>1013.2500</td>
<td>43.4860</td>
<td>0.96725</td>
<td>0.30</td>
<td>0.4E-4</td>
</tr>
<tr>
<td>/ 0.005/</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>1.0132</td>
<td>0.2947</td>
<td>1.03504</td>
<td>0.30</td>
<td>0.4E-4</td>
</tr>
<tr>
<td></td>
<td>533.7694</td>
<td>35.4555</td>
<td>0.98431</td>
<td>0.30</td>
<td>0.4E-4</td>
</tr>
<tr>
<td></td>
<td>1013.2500</td>
<td>42.1814</td>
<td>0.96725</td>
<td>0.30</td>
<td>0.4E-4</td>
</tr>
<tr>
<td>/</td>
<td>null record terminates table</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**SOLUH2SC**  Specify properties of water-H2S system with CO2 dependencies

This keyword enables the default aqueous phase property data to be overwritten. The data consists of a solubility table composed of a series of 2 records, each terminated by a slash. A null record, that is a record with no data before the terminating slash, must end this keyword.

The first line of record 2 should represent water at standard conditions when dissolved H$_2$S is at its minimum for a given CO$_2$ aqueous mole fraction. The simulator does not assume that the minimum H$_2$S in water is zero; if you want to specify that the amount of H$_2$S in water is zero at standard conditions then you should set item 2 to zero in each record 2 of the table.

**Record 1**

One item of data:

1. The CO$_2$ mole fraction.

**Record 2**

A table consisting of 5 columns of data.

1. Pressure
   
   Values should increase monotonically down the column.
   
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2. The dissolved H$_2$S solution ratio at the specified pressure and CO$_2$ concentration.
   
   This is expressed as the volume of H$_2$S at surface conditions per volume of water at surface conditions for a salt-saturated solution.
   
   UNITS: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)

3. The formation volume factor of water saturated with H$_2$S at the specified pressure and CO$_2$ concentration.
   
   UNITS: rm$^3$/sm$^3$ (METRIC), rb/stb (FIELD), rcc/scc (LAB), rm$^3$/sm$^3$ (PVT-M)

4. The viscosity of water saturated with H$_2$S at the specified pressure and CO$_2$ concentration.
   
   Note that Ezrokhi’s method may be used to generate the default water viscosity using item 164 of OPTIONS3. A default value of 0.3 cP will be used in the absence of this option switch.
   
   UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

5. The compressibility of water saturated with H$_2$S at the specified pressure and CO$_2$ concentration.
   
   UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

See also "CO$_2$/H$_2$S solution in the aqueous phase" in the ECLIPSE Technical Description.

**Example**

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---
<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>1.0132</td>
<td>0.3038</td>
<td>1.03504</td>
<td>0.30</td>
</tr>
<tr>
<td>533.7694</td>
<td>36.5522</td>
<td>0.98431</td>
<td>0.30</td>
<td>0.4E-4</td>
</tr>
<tr>
<td>1013.2500</td>
<td>43.4860</td>
<td>0.96725</td>
<td>0.30</td>
<td>0.4E-4</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>1.0132</td>
<td>0.2947</td>
<td>1.03504</td>
<td>0.30</td>
</tr>
<tr>
<td>533.7694</td>
<td>35.4555</td>
<td>0.98431</td>
<td>0.30</td>
<td>0.4E-4</td>
</tr>
<tr>
<td>1013.2500</td>
<td>42.1814</td>
<td>0.96725</td>
<td>0.30</td>
<td>0.4E-4</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
/ null record terminates table
SOLVCONC  Initial coal solvent concentration

This keyword may be used with the coal bed methane model (see "Coal bed methane model" in the ECLIPSE Technical Description).

The keyword should be followed by one real number for every matrix grid block, specifying the initial solvent concentration in the coal. The data field should be terminated by a slash (/).

UNITs: sm$^3$/m$^3$ (METRIC), Mscf/ft$^3$ (FIELD), scc/cc (LAB), sm$^3$/m$^3$ (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*). Data need only be supplied for the matrix cells, that is, the upper half of the grid.

Both SOLVCONC and SCVD are entirely optional. If neither of these keywords is present, the coal solvent concentration is taken as the equilibrium concentration for the corresponding fracture pressure and solvent fraction.

The initial coal solvent concentration should not be set above the initial equilibrium value; ECLIPSE issues a warning if it is. If the coal solvent concentration is below the equilibrium value, the non-reversible diffusion model is recommended (set item 2 in keyword DIFFCOAL to 0.0).

See also the GASCONC and SCVD keywords.

Example

With NDIVIX=8, NDIVIY=6 and NDIVIZ=10 specified in RUNSPEC keyword DIMENS

```plaintext
SOLVCONC
48*79.1
48*77.3
48*75.1
48*75.0
48*75.0 /
```
SOLVDIMS  Set dimensions for nested factorization solver for Pebi grids

This keyword instructs ECLIPSE that the solver operates on a given Cartesian box, but the data input is just for a list of active cells (supplied as a one dimensional list with all connections replaced by non-neighbor connections). The data is supplied by the FloGrid program.

The grid arrays need only be dimensioned according to the values in the DIMENS keyword. The SOLVDIMS keyword is only used for efficiency when the Nested Factorization solver is used, and is optional.

The number of cells $NX \times NY \times NZ$ specified in the DIMENS keyword must equal the number of nonzero items in the $NSOLX \times NSOLY \times NSOLZ$ array specified in the SOLVDIMS keyword. $NX$ should be the number of active cells, and $NY$ and $NZ$ should then both equal one. Typically $NSOLX \times NSOLY \times NSOLZ = NX \times NY \times NZ$.

The SOLVNUM array should be used to map from the input to solver order within ECLIPSE.

In a local grid, SOLVDIMS should be specified in the GRID section between the EXTFIN and ENDFIN keywords which define the local grid. For a global unstructured grid, SOLVDIMS should be specified in the RUNSPEC section.

The number of cells $NX \times NY \times NZ$ specified in the EXTFIN keyword must equal $NSOLX \times NSOLY \times NSOLZ$ specified in the SOLVDIMS keyword.

Example

```plaintext
RUNSPEC
  DIMENS
  1000 1 1 /
  SOLVDIMS
  10 10 10 /...
  GRID
  SOLVNUM
  ... /
```
SOLVDIRS  Overrides the solver principal directions

This keyword overrides the default directions for the linear solver. By default ECLIPSE chooses the principal direction to correspond to the direction of highest transmissibility. However, if there are persistent linear convergence problems it may be useful to try alternative principal directions.

The data consists of a two-character string, enclosed in quotes, terminated with a slash (/).

The options are:

- XY  X principal Y second and Z outer
- XZ  X principal Z second and Y outer
- YX  Y principal X second and Z outer
- YZ  Y principal Z second and X outer
- ZX  Z principal X second and Y outer
- ZY  Z principal Y second and X outer

ECLIPSE 100  To fix only the outer dimension (for matrix disking or the ECLIPSE 100 Parallel Option) set `-X' for the X direction, `-Y' for the Y direction or `-Z' for the Z direction. `-Z' is not permitted in parallel. To check the direction sequence set by the simulator, set DEBUG keyword item 4 to 1.

ECLIPSE 100  The SOLVDIRS keyword replaces item 3 of OPTIONS keyword that was used in pre-97A releases. But for back-compatibility with old datasets the OPTIONS switch still functions (in local grid refinement runs it overrides the solver directions for the global and all local grids).

When the local grid refinement option is used, the solver directions can be set independently for each local grid by including the SOLVDIRS keyword between the CARFIN / RADFIN / RADFIN4 / REFINE and ENDFIN pair. By default, the solver directions will be determined by ECLIPSE for each local grid unless set explicitly.

See "Solution of the linear equations" in the ECLIPSE Technical Description for further details.

Example

```
SOLVDIRS
'XZ' /
```
**SOLVENT**

**Enables the four-component solvent model**

This keyword indicates that the four-component solvent model is required (see "Solvent model" in the *ECLIPSE Technical Description*). The keyword should only be used with the other three phases (WATER, OIL, GAS) present.

The keyword has no associated data

*Note:* This keyword should not be used in conjunction with IMPES or DIMPES option.
**SOLVFRAC**  Initial solvent fraction in the gas phase

This keyword may be used with the coal bed methane model (see "Coal bed methane model" in the *ECLIPSE Technical Description*).

This keyword can be used to specify the initial solvent fraction in the gas phase, in runs in which the initial conditions are calculated by equilibration.

The keyword should be followed by one real number for every grid block, specifying the initial solvent fraction in the gas. The data field should be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest (from 1 to $NDIVIX$), followed by the Y axis index (from 1 to $NDIVIY$), and finally the Z axis index (from 1 to $NDIVIZ$). $NDIVIX$, $NDIVIY$ and $NDIVIZ$ are specified in RUNSPEC keyword *DIMENS*. Repeat counts may be used for repeated values (for example 115*0.18). Note that spaces must not be inserted on either side of the asterisk (*).

Data needs to be provided for the entire grid. However, the solvent fraction only applies to the cells in the fracture system. The equilibrium solvent concentration in the coal is calculated from the LANGSOLV / COALADS isotherm data and the prevailing fracture conditions.

See also the keyword *EQUIL* in the SOLUTION section.

**Example**

With $NDIVIX=10$, $NDIVIY=1$ and $NDIVIZ=2$ specified in RUNSPEC keyword *DIMENS*

```
SOLVFRAC
20*0.05 /
```
**SOLVNUM**  
Set map from user to solver order for Pebi grids

When the grid is supplied as a one-dimensional list of cells together with a collection of NNCs or MPFNNCs for an unstructured grid, the SOLVNUM array defines the mapping to solver order. The SOLVDIMS keyword should also be specified to inform ECLIPSE of the solver dimensions used when mapping to a Cartesian structured grid for the Nested Factorization linear solver. The data is supplied by the FloGrid program.

SOLVNUM should be followed by one integer for each grid cell in the current input box. For example, with SOLVDIMS of \( NSOLX \times NSOLY \times NSOLZ \):

\[
SOLVNUM(IX, IY, IZ) = ISX + (ISY - 1) \times NSOLX + (ISZ - 1) \times NSOLX \times NSOLY
\]

where \((ISX, ISY, ISZ)\) is the solver position in the box \([1, NSOLX] \times [1, NSOLY] \times [1, NSOLZ]\).

In a local grid, the DIMENS data is supplied by the EXTFIN keyword which defines the extent of the unstructured local grid. The SOLVDIMS keyword should be specified between the EXTFIN and ENDFIN keywords. Then SOLVNUM should be specified between the EXTFIN and ENDFIN pair or a matching REFIN and ENDFIN pair.

**Note:** The MPFA option is currently not available for use with this keyword. As a consequence, ECLIPSE 300 will ignore any MPFNNC read in.

**Examples**

**Example 1**

There are 1000 x 1 x 1 cells, mapped to an array of dimension 10 x 10 x 10 in the solver. Gaps are filled by inactive cells.

```
RUNSPEC  
.. . . .
DIMENS  
1000 1 1  
/
SOLVDIMS  
10 10 10  
/
GRID  
...  
SOLVNUM  
... ..  
/
```

**Example 2**

In the local grid ‘NWB’, SOLVDIMS and SOLVNUM are both specified after the EXTFIN keyword.

```
GRID  
.. . . .  
EXTFIN  
‘NWB’ 1000 1 1 15 9 10 65 34 2  
/
SOLVDIMS  
10 10 10  
/
SOLVNUM  
...  
/
ENDFIN
```
**SOLWNUM**  

**Solid wettability region numbers**

This keyword specifies the saturation table numbers used in the **SOLID** model to calculate the relative permeabilities and capillary pressures when the adsorbed solid concentration is high. Alternatively, the **SATNUM** keyword is used for the saturation functions when the adsorbed solid concentration is low. The **SOLWTAB** keyword determines the interpolation factor between these extremes.

The keyword should be followed by one integer for every grid block in the current input box, specifying the saturation function region numbers for the areas of high concentration of adsorbed solid. The region number should not be less than 1 or greater than **NTSFUN** (see **RUNSPEC** section keyword **TABDIMS**). The data must be terminated by a slash (/).

The saturation functions are specified using the **SWFN**, **SOF2**, **SOF3** or **SWOF** keywords.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

**Note:** If end-point scaling has been specified via the **ENDSCALE** option, both the high and low adsorbed concentration tables (specified via **SOLWNUM** and **SATNUM** respectively) will use the same set of end-points, for example, **SWL**, **SWCR**, **SWU** and related end-points.

**Example**

With **NTSFUN=4** specified in keyword **TABDIMS**; **NDIVIX=10, NDIVIY=10 and NDIVIZ=1** specified in **RUNSPEC** keyword **DIMENS**; and no input **BOX** set

```
SATNUM
  50*1 50*2 /
SOLWNUM
  50*3 50*4 /
```
**Solid wettability table**

The data comprises NTSFUN (item 1 of keyword TABDIMS) tables that define an interpolation parameter as a function of the adsorbed solid concentration. The parameter is used to interpolate between two sets of saturation functions, representing two different wettability conditions, defined by SATNUM and SOLWNUM. Oil and water relative permeability and capillary pressure saturation functions are interpolated. The gas saturation functions are not affected by solid wettability and only the SATNUM tables are used for gas permeability and capillary pressure.

Each table consists of 2 columns of data and should be terminated by a slash (/):

1. Concentration of adsorbed solid
   - The values should increase monotonically down the column.
   - **UNITS:** kg/kg (METRIC), lb/lb (FIELD), gm/gm (LAB), kg/kg (PVT-M)

2. Interpolation parameter
   - The value should not be greater than 1.0 or less than 0.0 A value of 0 means the SATNUM saturation functions and a value of 1 means the SOLWNUM saturation functions will be used.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (item 3 of keyword TABDIMS).

**Note:** NTSFUN tables must be supplied. However only SOLWTAB tables referenced by SATNUM are used. If any table after the first is defaulted, then that table is copied from the previous table.

The keyword can only be used with the SOLID model. The ROCKDEN keyword must be supplied, to allow the adsorbed solid concentration to be determined. The keyword can be used with standard saturation functions and can also be used with the oil-water miscibility model which is described in the "Surfactant model" in the ECLIPSE Technical Description. However, it cannot be used with other saturation function options, such as hysteresis, ODD3P and IKU3P.

**Example**

With NTSFUN=4 and NSSFUN>=4

```
SOLWTAB
0   0.0
20  0.2
600 0.3
1800 0.6/
    / default second table to the first table
0   0.0
2   0.0 / dummy table corresponding to a SATNUM table value
/ default the fourth table to the third table
```

---

**Keywords**

2112
**SOLWW**

**Solid wettability weighting factor**

This keyword can be used with the SOLID model to specify a weighting factor for use in the solid wettability calculation. By default, the component molecular weight $MW$ is used to calculate the adsorbed solid concentration, which can then be used with the SOLWTAB keyword to determine wettability effects. When there is more than one solid component, this keyword can be used to multiply the molecular weight, allowing different components to contribute differently to the wettability.

The keyword should be followed by $N_c$ values, one for each hydrocarbon component. However, only values for solid components are significant.

- **UNITS:** kg/kg-M (METRIC), lb/lb-M (FIELD), gm/gm-M (LAB), kg/kg-M (PVT-M)
- **DEFAULT:** 1.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

### Example

In a run with three solid components, the full weight of the first two solid components will be included in the wetting-solid concentration, while the third solid component will have no effect on the wettability:

```
CVTYPE 1* 1* SOL SOL SOL /
MW 18.0 32.0 45.6 78.0 88.0 98.0 /
SOLWW 1* 1* 1* 1.0 0.0 /
```
SOMGAS  Minimum oil saturation for the STONE I model

This keyword is optional, and specifies the minimum oil saturation used in the STONE I three-phase relative permeability model, as a function of the gas saturation. The data will be ignored if the STONE1 keyword has not been used.

The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) tables of minimum oil saturation. Each table consists of two columns of data, terminated by a slash (/):

1. The gas saturation.
   Values should increase down the column.
   - The first gas saturation should equal the connate gas saturation in the corresponding SGFN table (which is usually zero).
   - The last gas saturation in the table should equal 1 – S_{wco} - S_{ogcr} where S_{wco} is the connate water saturation and is the residual oil-to-gas saturation, in the corresponding saturation tables. (S_{wco} is the lowest saturation in the SWFN table and S_{ogcr} is the largest oil saturation with k_{rog} of zero, in column 3 of the SOF3 keyword.)

2. The corresponding minimum oil saturation.
   - The minimum oil saturation at connate gas (first point) should not exceed S_{owcr}, the residual oil-to-water saturation in the corresponding SOF3 table. (S_{owcr} is the largest oil saturation with k_{row} of zero, in column 2 of the SOF3 keyword.)
   - The minimum oil saturation at 1 – S_{wco} - S_{ogcr} (last point) should not exceed S_{ogcr}, the residual oil-to-gas saturation in the corresponding SOF3 table.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see item 3 of keyword TABDIMS).

The entire table may be defaulted, provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

An alternative keyword SOMWAT can be used to input the minimum oil saturation; SOMWAT tabulates the minimum oil saturation against the water saturation rather than the gas saturation. Either keyword can be used to describe the same minimum oil isoperm. However, the actions of the two keywords are slightly different at oil saturations above the minimum. Using the SOMGAS keyword implies that the SOM is looked up at the prevailing gas saturation. Both keywords cannot be used in the same run.

The alternative family of saturation keywords (SWOF, SGOF, SLGOF) can be used to input the relative permeability data. In this case the connate water saturation S_{wco} and the residual oil-to-water saturation S_{owcr} are determined from the SWOF table and the residual oil-to-gas saturation S_{ogcr} is determined from the SGOF or SLGOF table.

Note: The consistency requirements described above refer to the table end-points irrespective of whether or not end-point scaling has been specified.

Refer to "Saturation functions" in the ECLIPSE Technical Description for more information.

Example

With NTSFUN=1 and NSSFUN ≥6:

Keywords
SWF3

-- Sw  Kr  Pc
  0.25  0.0  0.0
  0.7   1.0  0.0  /

SOF3

-- So  Krow  Krog
  0.1   0.0  0.0
  0.31  0.0  0.5
  0.75  1.0  1.0  /

SOMGAS

  0.0  0.31
  0.1  0.2
  0.2  0.1
  0.3  0.05
  0.4  0.01
  0.5  0.05
  0.65  0.1  /

1-SWCO-SOCGR (1 - 0.25 - 0.1)
Minimum oil saturation for the STONE I model

This keyword is optional, specifies the minimum oil saturation used in the STONE I three-phase relative permeability model, as a function of the water saturation. The data will be ignored if the STONE1 keyword has not been used.

The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) tables of minimum oil saturation. Each table consists of two columns of data, and is terminated by a slash (/).

1. The water saturation.

Values should increase down the column.

The first water saturation should equal the connate water saturation in the corresponding SWFN table. (The connate water saturation is the lowest saturation in the SWFN table.)

The last water saturation in the table should equal to 1 – Sgco - Sowcr where:

Sgco is the connate gas saturation

Sowcr is the residual oil-to-water saturation in the corresponding SOF3 table.

(Sowcr is the largest oil saturation with krow of zero, in column 2 of the SOF3 keyword.

2. The corresponding minimum oil saturation.

The minimum oil saturation at connate water (first point) should not exceed Sogcr, the residual oil-to-gas saturation in the corresponding SOF3 table. (Sogcr is the largest oil saturation with krog of zero, in column 3 of the SOF3 keyword.)

The minimum oil saturation at 1 – Sgco - Sowcr (last point) should not exceed Sowcr, the residual oil-to-water saturation in the corresponding SOF3 table.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see item 3 in keyword TABDIMS).

The entire table may be defaulted, provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

An alternative keyword SOMGAS can be used to input the minimum oil saturation; SOMGAS tabulates the minimum oil saturation against the gas saturation rather than the water saturation. Either keyword can be used to describe the same minimum oil isoperm. However, the actions of the two keywords are slightly different at oil saturations above the minimum. Using the SOMWAT keyword implies that the SOM is looked up at the prevailing water saturation. Both keywords cannot be used in the same run.

The alternative family of saturation keywords (SWOF, SGOF, SLGOF) can be used to input the relative permeability data. In this case the connate water saturation Swco and the residual oil-to-water saturation Sowcr are determined from the SWOF table and the residual oil-to-gas saturation Sogcr is determined from the SGOF or SLGOF table.

Note: The consistency requirements described above refer to the table end-points irrespective of whether or not end-point scaling has been specified.

See "Saturation functions" in the ECLIPSE Technical Description for more information.
Example

With **NTSFUN=1** and **NSSFUN \geq 6**: 

```
SWFN
-- Sw  Kr   Pc
  0.25 0.0  0.0
  0.7  1.0  0.0 /

SOF3
-- So  Krow  Krog
  0.1  0.0  0.0
  0.31 0.0  0.5
  0.75 1.0  1.0 /

SOMWAT
  0.25 0.1  Connate water
  0.3  0.08
  0.4  0.06
  0.5  0.1
  0.6  0.2
  0.69 0.3  1 - SOWCR
/
```
Residual oil saturation

This keyword should be used in conjunction with EOS compositional simulations. It specifies a residual oil saturation that does not vaporize. Residual oil could be modeled in a number of ways: by using heavy oil components that do not vaporize (though these would be mobile); by using critical oil saturations (though the oil would be volatile); by using SOLID components; or by using dual porosity (DUALPORO). The SOR model is intended as a simple alternative to these.

SOR is a replacement for SORMW1 and works in a different way. In the SOR model, the hydrocarbons are divided into two: residual oil and mobile hydrocarbons. The residual oil composition remains fixed throughout the simulation. It is treated as a separate phase that does not flow and does not contribute to the flash. It does contribute to the cell oil saturation and calculations of saturation functions such as relative permeabilities. The residual oil phase can be thought of as being trapped in pores, separate from the mobile hydrocarbon in the cell. The residual oil has only two properties: compressibility and saturation.

In the residual oil model, the oil saturation is divided into two parts, the mobile oil saturation and the residual oil saturation:

\[ SOIL = SOILM + SOILR \]

The residual oil saturation given by this keyword is the fraction of the pore volume that is residual oil. It is not the fraction of the hydrocarbon volume (as used by some other authors).

Related keywords

Three optional keywords can be used with SOR.

SOILR  By default the residual oil saturation is set by the SOR keyword, with the same value for all cells in each saturation region. However, the SOILR keyword can be used to override the SOR values and specify the residual oil saturation on a cell-by-cell basis.

ROMF  By default the residual oil composition is set equal to the initial oil composition in each cell, but the ROMF keyword can be used to specify the composition directly.

SOROPTS  By default the residual oil saturation in a cell remains constant throughout the simulation. However a more physically realistic compressibility model can be set using the SOROPTS keyword.

Initialization

The way the initial residual oil is calculated varies, depending on whether the ROMF keyword is used.

If the ROMF keyword is used to set the residual oil composition, then the mobile oil composition is set equal to the original cell composition, and mixed with the residual oil in proportion \(1 - SOILR_o:SOILR_o\), where \(SOILR_o\) is the initial residual oil saturation, equal to the SOR or SOILR value.

If the ROMF keyword is not used, the oil in the cell is partitioned into residual oil and mobile oil. If there is not enough oil in the cell, the initial residual oil saturation is reduced to the initial oil saturation \(SOILR_o < SOIL_o\). The residual oil composition will be equal to the initial oil composition.

Simulation

The number of residual oil moles in a cell stays constant throughout the simulation, which means the concentration will change as the pressure changes.
The mobile hydrocarbons are flashed in the usual way to determine the phase split and phase properties (such as density and viscosity). The phase saturations are then modified to include the residual oil. Therefore relative permeabilities and capillary pressures will be based on the total oil saturation in the cell. It is recommended that the oil relative permeability is set to zero for oil saturations below the residual oil saturation.

**Output**

Reported oil saturations $SOIL$ represent the total oil saturation, including both residual and mobile oil. Other fluid properties (such as oil phase density, viscosity, $XMF$) are for the mobile hydrocarbon only. In the case where there is no mobile oil, the cell hydrocarbon state will be single phase gas, although the cell may contain residual oil and water. Apart from saturation, the oil phase properties for cells with no mobile hydrocarbon are meaningless; this is also true of oil phase properties for gas-filled cells in normal simulations, but may seem surprising since these cells can have a non-zero (residual) oil saturation.

Fluid in place calculations are performed using both the residual oil and the mobile hydrocarbons.

The composition of the residual oil and the saturations of residual and mobile oil can be output to the print file $RPT SOL$, $RPT SCHED$, the restart file $RPTRST$, and the summary file, using the following mnemonics and keywords:

<table>
<thead>
<tr>
<th></th>
<th>$RPT SOL/RPT SCHED/RPTRST$</th>
<th>SUMMARY vector</th>
</tr>
</thead>
<tbody>
<tr>
<td>Residual oil composition</td>
<td>ROMLS</td>
<td>BROMLS</td>
</tr>
<tr>
<td>Residual oil saturation</td>
<td>SOILR</td>
<td>BSOILR</td>
</tr>
<tr>
<td>Residual oil compressibility (see $SOROPTS$)</td>
<td>COILR</td>
<td>BCOILR</td>
</tr>
<tr>
<td>Mobile oil saturation</td>
<td>SOILM</td>
<td>BSOILM</td>
</tr>
</tbody>
</table>

*Table 3.49: Residual and mobile oil output keywords*

**Keyword data**

The data comprises $NTSFUN$ (see item 1 of keyword $TABDIMS$ in the $RUNSPEC$ section) values, each terminated by a slash. Each value represents the residual oil saturation in each saturation region and should be between 0 and 1. Values close to 1 may cause convergence issues, since the effective pore volume (that is that available to flow) is proportional to $1-SOILR$.

**Example**

With $NTSFUN=2$

```
SOR .1 / .15 /
```
SORBFRAC  Initial adsorption fraction

This keyword may be used with the coal bed methane model (see "Coal bed methane model" in the ECLIPSE Technical Description).

This keyword can be used to specify that only a fraction of the calculated equilibrium gas contents is actually present in the coal matrix.

**Note:** The same fraction is applied to all components.

The keyword should be followed by one real number for every grid block, specifying the fraction used. The data field should be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*0.18). Note that spaces must not be inserted on either side of the asterisk (*).

Only the values given for the matrix cells are used, so it is sufficient to supply NDIVIX * NDIVIY * NDIVIZ/2 values. The molar densities in the coal is calculated from the LANGMEXT isotherm data and the prevailing matrix conditions and then multiplied with the scaling fraction.

The keyword is entirely optional, if not specified, or defaulted values are used, the value of unity is assumed.

**DEFAULT:** 1.0

Values should be in the range [0,1] but values larger than unity are not flagged as an error, but a message is given. Values larger than unity means that the initial coal gas contents are not in equilibrium (unless the sorption pressure is sufficiently low).

See also the keyword EQUIL in the SOLUTION section.

**Example**

With NDIVIX=10, NDIVIY=1 and NDIVIZ=2 specified in RUNSPEC keyword DIMENS:

```
SORBFRAC
  5* 5*0.95 /
```

The first 5 cells are defaulted to 100%, while for the 5 next cells 95% of the adsorption capacity as given by the Langmuir isotherm is initialized in the coal matrix.
**SORBPRES**  
**Initial adsorption pressure**

This keyword may be used with the coal bed methane model (see "Coal bed methane model" in the *ECLIPSE Technical Description*).

The keyword specify the initial sorption pressure. The sorption pressure overwrites the computed matrix pressure, as given by equilibration, and is then used to set the initial coal gas contents.

The keyword should be followed by one real number for every grid block, specifying the pressure for each cell in the matrix. The data field should be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*1009). Note that spaces must not be inserted on either side of the asterisk (*).

Only the values given for the matrix cells are used, so it is sufficient to supply NDIVIX * NDIVIY * NDIVIZ /2 values.

The molar densities of the coal are calculated from the LANGMEXT isotherm data, the input gas mole fractions and the sorption pressure.

The keyword is entirely optional and if not specified, or defaulted values are used, the computed value from equilibration or the explicitly set pressure value is used.

See also the keyword EQUIL in the SOLUTION section.

**UNITS**: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

**Example**

With NDIVIX=10, NDIVIY=1 and NDIVIZ=2 specified in RUNSPEC keyword DIMENS:

```
SORBPRES
  5* 5*1009.0 /
```

The first 5 cells are unaltered, while the 5 next cells are set to a sorption pressure of 1009.0. This is used when computing the adsorption capacity as given by the Langmuir isotherm.
Residual oil saturation for compositional

This keyword has been replaced by the SOR keyword.

This keyword should be used in conjunction with the ENDScale keyword. It allows the user to set a residual oil saturation that does not vaporize due for example to continuous gas injection.

The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) values, each terminated by a slash. Each value represents the residual oil saturation in each saturation region.

The residual oil saturation should not be higher than the critical oil in gas saturation (SOGCR). If it is the case, then the residual oil saturation is reset to SOGCR.

Example

With NTSFUN=2

```
SORMW1
.1 / .15 /
```

Note: The composition of the residual oil is set as a fraction of the initial oil composition after equilibration. This fraction is not flashed, and is used later on to calculate the oil molar as a function of pressure. In this version of ECLIPSE the composition of the residual oil does not change with time. Note that in some cases tuning may be needed (for example reducing the maximum timestep using the TSCRIT keyword) to adjust the timestep for a better performance. ENDScale
Residual oil model options

This keyword specifies options for the residual oil model. See the SOR keyword. Only one option is currently available: the compressibility model for the residual oil.

The residual oil saturation is the ratio of the residual oil volume in a cell to the pore volume:

\[ SOILR = \frac{VOILR}{PORV} \]

Since the residual oil composition remains constant, the volume of residual oil depends only on pressure. The residual oil saturation therefore depends only on pressure, given by:

\[ \frac{1}{SOILR} \cdot \frac{dSOILR}{dP} = -COILR \cdot \frac{1}{PORV} \cdot \frac{dPORV}{dP} \]

where \( COILR \) is the residual oil compressibility:

\[ COILR = -\frac{1}{VOILR} \cdot \frac{dVOILR}{dP} \]

Three compressibility models are available:

**Constant SOR model**

`ZERO`  \[ \frac{1}{SOILR} \cdot \frac{dSOILR}{dP} = 0 \]

The residual oil saturation remains constant throughout the simulation. This is the simplest model, but not physical, since it implies the residual oil has a negative compressibility. As the pressure increases and the pore volume increases, the volume of residual oil must increase at the same rate.

**Constant compressibility model**

`CONSTANT`  \[ COILR = CONSTANT \]

The initial compressibility of the residual oil is calculated and used throughout the simulation.

**Flash compressibility model**

`FLASH`

The residual oil volume is determined by flashing the residual oil composition at the cell pressure. However, if the cell pressure drops below the bubble point of the residual oil, then the calculation will switch to the ‘CONSTANT’ method for that cell, using the previous single-phase value of the residual oil compressibility in that cell. The calculation will revert to the ‘FLASH’ method if the pressure increases above the residual oil bubble point.

This keyword is followed by one item, specifying the compressibility model.

DEFAULT: ZERO

Example

```
SOROPTS
FLASH  /
```
SORWMIS  Miscible residual oil saturation tables

The data comprises NTMISC (item 1 in keyword MISCIBLE in the RUNSPEC section) tables of miscible residual oil saturation versus water saturation, each terminated by a slash. "Miscible flood modeling" in the ECLIPSE Technical Description contains details of how the effective residual oil saturation tables are used in the calculation of the gas and oil relative permeabilities and PVT properties. See also "Solvent model" in the ECLIPSE Technical Description for further information when the four-component solvent model is employed.

The SORWMIS keyword can only be used in runs which use the miscible flood option (keyword MISCIBLE in the RUNSPEC section) and where oil, water and gas are all present. For miscible flood simulations involving the water phase the SORWMIS keyword may be omitted, in which case the program assumes a miscible residual oil saturation of zero.

Each table consists of 2 columns of data:

1. The water saturation.
   Values should be between 0 and 1 and should increase monotonically down the column.

2. The corresponding miscible residual oil saturation.
   Values should be between 0 and 1 and should be level or increasing down the column

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSMISC (item 2 in keyword MISCIBLE).

**Example**

With NTMISC=2 and NSMISC ≥ 3:

```
SORWMIS
.00  .00
.30  .00
1.00  .00 /
.00  .00
.25  .07
.80  .11 /
```

---

**Keywords**

ECLIPSE 100  ECLIPSE 300  SPECIAL
RUNSPEC  GRID  EDIT
x  PROPS  REGIONS  SOLUTION  SUMMARY
SCHEDULE

ECLIPSE Reference Manual

2124
The **SOWCR** and **ISOWCR** keywords specify the critical oil-in-water saturation (that is the largest oil saturation for which the oil relative permeability is zero in an oil-water system) within each grid cell. The oil flow across each grid face in an oil-water system is calculated from a transformed oil-water relative permeability curve obtained by linearly scaling the tabulated oil-water relative permeability data between the new critical oil-in-water saturation defined using an **SOWCR** keyword, and the new maximum oil saturation (see under keywords **SWL**, **SGL** in the **PROPS** section). The **SOWCR** keyword also permits scaling of the oil-water relative permeability table used in computing the flow of oil between grid cells and well connections.

Seven keywords are provided to allow the critical oil-in-water saturation for each grid cell to be scaled independently.

These keywords are:

- **SOWCR**  
  Critical oil-water saturation for the non-directional saturation function tables

- **SOWCRX**  
  Critical oil-water saturation for the +X face of each grid cell

- **SOWCRX-**  
  Critical oil-water saturation for the -X face of each grid cell

- **SOWCRY**  
  Critical oil-water saturation for the +Y face of each grid cell

- **SOWCRY-**  
  Critical oil-water saturation for the -Y face of each grid cell

- **SOWCRZ**  
  Critical oil-water saturation for the +Z face of each grid cell

- **SOWCRZ-**  
  Critical oil-water saturation for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the critical oil-in-water saturations for flows in the +R, -R, +T, -T, +Z, -Z directions respectively.

None of the **SOWCR** keywords should be used if either oil or water is absent from the model. The keyword **ENDSCALE** should be specified in **RUNSPEC**.

If both directional and irreversible end-point scaling is requested, using switches ’DIRECT’ and ’IRREVERS’ in the **ENDSCALE** keyword in **RUNSPEC**, then the critical oil-water saturations for oil flow across the +X and -X faces of a grid cell are set using the keywords **SOWCRX**, **SOWCRX-**. Similarly **SOWCRY**, **SOWCRY-** are used for the +Y, -Y faces and the keywords **SOWCRZ**, **SOWCRZ-** for the +Z, -Z faces. Scaling of the well connection relative permeability tables for any well connections in the grid cell is performed using the **SOWCR** keyword.

When the end-point scaling is requested to be directional and reversible, using the switch ’DIRECT’ but not ’IRREVERS’ in **ENDSCALE**, the keyword **SOWCRX** defines the critical oil-water saturations for the +X and -X faces of each grid cell. Keyword **SOWCRY** is used for both the +Y, -Y faces and the keyword **SOWCRZ** for both the +Z, -Z faces of each grid cell. Scaling of well connection relative permeability tables uses data entered under the **SOWCR** keyword.

If **ENDSCALE** is specified in **RUNSPEC** without the switch ’DIRECT’, then none of the directional forms of the **SOWCR** keyword are allowed. The **SOWCR** keyword then defines the new critical oil-water saturation for flow out of each face of the grid cell and for oil flows between the grid cell and well connections.
Each SOWCR keyword should be followed by one real number for each grid block in the current input box specifying the critical oil-water saturation within the grid block. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If any of the keywords SOWCR, SOWCRX, SOWCRX-, SOWCRY, SOWCRY-, SOWCRZ, SOWCRZ- are omitted in a model where they could be accepted, then the critical oil-in-water saturation for an oil relative permeability calculation defaults to the value used in the appropriate relative permeability table.

**ECLIPSE 100**

If the Hysteresis model (switch 'HYSTER' in keyword SATOPTS) is active then the SOWCR data scales the drainage curve. The imbibition curve can be scaled by using the equivalent ISOWCR (ISOWCRX, ISOWCRX- etc.) keyword.

**ECLIPSE 300**

If the ODD3P relative permeability and capillary pressure hysteresis model has been selected, the primary end-point scaling is specified using the PSORG, PSGRO, PSORW, PSWRO, PSGRW and PSWRG keywords and the hysteresis end-point scaling is specified using the HSORG, HSGRO, HSORW, HSWRO, HSGRW and HSWRG keywords.

See also keywords SGL, SWL, ENPTVD in the PROPS section.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the endpoint scaling procedure.

**Note:** When consistency warning messages involving SOWCR are reported by ECLIPSE, in some cases this may be referred to using the alternative name SOCRW. (It should be noted that this is not accepted as an alternative keyword name.)

### Example

```
--------RUNSPEC section
DIMENS 1 7 3 /
ENDSCALE /
--------PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX 1 1 2 7 1 3
SOWCR 5*0.21 10*0.19 3*0.18 /
```

Keywords

2126
**SPECGRID**

**Specification of grid characteristics**

This keyword repeats the specification of dimensions and other properties describing the simulation grid, as defined in the RUNSPEC section.

The SPECGRID keywords use is optional. The data specified in this keyword is not actually used. It only serves a checking function.

It is normally output by grid pre-processors (such as the FloGrid program) so that the program can verify that the same properties are used in both data sets. A error message is issued if agreement between the program and the pre-processor is not reached.

Following the keyword line there is a single record containing the following items, and ending with a slash (/):

1. The number of grid blocks in the I (X or radial) direction (NDIVIX, specified in RUNSPEC keyword **DIMENS**)
   
   DEFAULT: 1

2. The number of grid blocks in the J (Y or theta) direction (NDIVIY, specified keyword **DIMENS**)
   
   DEFAULT: 1

3. The number of grid blocks in the K (Z or depth) direction (NDIVIZ, specified keyword **DIMENS**)
   
   DEFAULT: 1

4. The number of reservoirs (NUMRES)
   
   Each reservoir has its own coordinate system.
   
   DEFAULT: 1

5. Type of coordinates
   
   T Cylindrical (radial) coordinates.
   
   F Cartesian coordinates.
   
   DEFAULT: F

**ECLIPSE 100 only**

4. The number of reservoirs (NUMRES)
   
   Each reservoir has its own coordinate system.
   
   DEFAULT: 1

**ECLIPSE 300**

**Note:** Only the first 3 items have any use in this simulator. The 4th and 5th items can be read in, but are ignored.

See also keywords **DIMENS, NUMRES, COORDSYS** and **COORD**.

**Example**

```
SPECGRID
5 3 4 2 T /
```

---

**Keywords**

2127
Oil component specific heat

This keyword can only be used with the THERMAL option.

SPECHA specifies the first coefficient of the oil component liquid specific heats, for each EoS region.

For multi-component simulations, the oil phase molar enthalpy is the mole fraction weighted sum of the component enthalpies:

\[ h_o = \sum_{c=1}^{N_c} x_c \cdot h_c \cdot mw_c \]  

Eq. 3.176

where

- \( x_c \) is the mole fraction of component \( c \) in the oil phase, and
- \( mw_c \) is the component molecular weight.

The component liquid enthalpy \( h_c \) has the form

\[ h_c = C_{p1c} \cdot (T-T_{st}) + \frac{1}{2} \cdot C_{p2c} \cdot (T-T_{st})^2 \]  

Eq. 3.177

where

- \( C_{p1c} \) is the first coefficient of the component liquid specific heat entered with this keyword
- \( C_{p2c} \) is the second coefficient of the component liquid specific heat entered with the SPECHB keyword, and
- \( T_{st} \) is the standard temperature defined with keyword STCOND.

**Note:** It is common in thermal simulation to specify the component liquid specific heats with keyword SPECHA. Then to obtain the enthalpies of gaseous components, a heat of vaporization for each component is supplied with keyword HEATVAP. Alternatively, one can specify gas component enthalpies with keyword SPECHG and obtain the liquid enthalpies by again specifying the component heats of vaporization with keyword HEATVAP. Either SPECHA or SPECHG must be set for each run. Only two of the three keywords SPECHG, SPECHA and HEATVAP should be specified. For further information please see "Enthalpy" in the ECLIPSE Technical Description.

This keyword should only be used for thermal runs with volatile hydrocarbon components. For DEADOIL runs, use the OILSPECH keyword.

The data record contains one value for each component:

1. liquid hydrocarbon component specific heats, \( N_c \) values, one for each component

   **UNITS:** kJ/kg/K (METRIC), Btu/lb/°R (FIELD), J/gm/K (LAB), kJ/kg/K (PVT-M)

   **DEFAULT:** 0.0
If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of \texttt{TABDIMS}. The use of multiple EoS regions is discussed in "Equations of state regions" in the \textit{ECLIPSE Technical Description}.

\section*{Example}

For 4 components

\begin{verbatim}
SPECHA
0.55 0.57 0.58 0.61 /
\end{verbatim}
SPECHB

Oil component specific heat

This keyword can only be used with the THERMAL option.

SPECHB specifies the second coefficient of the oil component liquid specific heats, for each EoS region.

For multi-component simulations, the oil phase molar enthalpy is the mole fraction weighted sum of the component enthalpies:

$$h_o = \sum_{c=1}^{N_c} x_c \cdot h_c \cdot mw_c$$  \hspace{1cm} \text{Eq. 3.178}

where

- $x_c$ is the mole fraction of component $c$ in the oil phase, and
- $mw_c$ is the component molecular weight.

The component liquid enthalpy $h_c$ has the form

$$h_c = C_{p_{1c}} \cdot (T-T_{st}) + \frac{1}{2} \cdot C_{p_{2c}} \cdot (T-T_{st})^2$$  \hspace{1cm} \text{Eq. 3.179}

where

- $C_{p_{1c}}$ is the first coefficient of the component liquid specific heat entered with the SPECHA keyword,
- $C_{p_{2c}}$ is the second coefficient of the component liquid specific heat entered with this keyword, and
- $T_{st}$ is the standard temperature defined with keyword STCOND.

**Note:** It is common in thermal simulation to specify the component liquid specific heats with keyword SPECHA. Then to obtain the enthalpies of gaseous components, a heat of vaporization for each component is supplied with keyword HEATVAP. Alternatively, one can specify gas component enthalpies with keyword SPECHG and obtain the liquid enthalpies by again specifying the component heats of vaporization with keyword HEATVAP. Either SPECHA or SPECHG must be set for each run.

This keyword should only be used for thermal runs with volatile hydrocarbon components. For DEADOIL runs, use the OILSPECH keyword.

The data record contains one value for each component:

1. liquid hydrocarbon component specific heats, $N_c$ values, one for each component

   **UNITS:** kJ/kg/K (METRIC), Btu/lb/°R (FIELD), J/gm/K (LAB), kJ/kg/K (PVT-M)

   **DEFAULT:** 0.0

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.
**Example**

For 4 components

```
SPECHB
0.005 0.007 0.008 0.006 /
```
**SPECHEAT**  **Fluid specific heat data**

This keyword should only be used for runs with the Temperature option active (see keyword **TEMP**).

The data comprises NTPVT (see keyword **TABDIMS**) tables, defining the specific heats of oil, water and gas as a function of temperature.

Each table consists of 4 columns of data and is terminated by a slash (/):

1. The temperature.
   The values should increase monotonically down the column.
   **UNITS:** °C (**METRIC**), °F (**FIELD**), °C (**LAB**), °C (**PVT-M**)

2. The corresponding oil specific heat.
   **UNITS:** kJ/kg/K (**METRIC**), Btu/lb/°R (**FIELD**), J/gm/K (**LAB**), kJ/kg/K (**PVT-M**)

3. The corresponding water specific heat.
   **UNITS:** kJ/kg/K (**METRIC**), Btu/lb/°R (**FIELD**), J/gm/K (**LAB**), kJ/kg/K (**PVT-M**)

4. The corresponding gas specific heat.
   **UNITS:** kJ/kg/K (**METRIC**), Btu/lb/°R (**FIELD**), J/gm/K (**LAB**), kJ/kg/K (**PVT-M**)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see keyword **TABDIMS**).

See also the **SPECROCK** keyword, and the "Temperature option" in the **ECLIPSE Technical Description**.

**Example**

With NTPVT=1 and NPPVT ≥2:

```
SPECHEAT
  0.0  0.5  1.5  0.5
  300.0  0.5  1.5  0.5
/
```

Keywords

2132
SPECHG

Gas component specific heat

This keyword can only be used with the THERMAL option.

SPECHG specifies the first coefficient of the gas component specific heat, for each EoS region.

For multi-component simulations, the gas phase molar enthalpy is a mole fraction weighted sum of the component enthalpy values:

\[ h_g = \sum_{c=1}^{N_c} y_c \cdot h_c \cdot m_w_c \]  

Eq. 3.180

where

\[ y_c \] is the mole fraction of component \( c \) in the gas phase, and \n
\[ m_w_c \] is the component molecular weight.

The component gas phase enthalpy \( h_c \) has the form

\[ h_c = h_{cs} + C_{p1c} \cdot (T - T_{st}) + \frac{1}{2} \cdot C_{p2c} \cdot (T - T_{st})^2 \]  

Eq. 3.181

where

\[ T_{st} \] is the standard temperature defined with keyword STCOND,

\[ h_{cs} \] is the heat of vaporization at \( T_{st} \), defined by HEATVAPS,

\[ C_{p1c} \] is the first coefficient of the component gaseous specific heat entered with this keyword, and

\[ C_{p2c} \] is the second coefficient of the component gaseous specific heat entered with the SPECHH keyword.

Note: It is common in thermal simulation to specify the component liquid specific heats with keyword SPECHA. Then to obtain the enthalpy values of gaseous components, a heat of vaporization for each component is supplied with keyword HEATVAP. Alternatively, one can specify gas component enthalpy values with this keyword and obtain the liquid enthalpy values by again specifying the component heats of vaporization with keyword HEATVAP. Either SPECHA or SPECHG must be set for each run. Only two of the three keywords SPECHG, SPECHA and HEATVAP should be specified. For further information please see “Enthalpy” in the ECLIPSE Technical Description.

This keyword should only be used for thermal runs with volatile gas. For DEADOIL runs the hydrocarbon components do not appear in the gas phase.

The data record contains one value for each component:

1. gas hydrocarbon component specific heats, \( N_c \) values, one for each component

   UNITS: kJ/kg/K (METRIC), Btu/lb/°R (FIELD), J/gm/K (LAB), kJ/kg/K (PVT-M)

   DEFAULT: 0.0
If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of `TBDIMS`. The use of multiple EoS regions is discussed in "Equations of state regions" in the *ECLIPSE Technical Description*.

**Example**

For four components

```
SPECHG
0.5E-2 0.5E-2 0.5E-2 0.6E-2 /
```

**Keywords**

2134
SPECHH specifies the second coefficient of the gas component specific heat, for each EoS region.

For multi-component simulations, the gas phase molar enthalpy is a mole fraction weighted sum of the component enthalpy values:

\[ h_g = \sum_{c=1}^{N_c} y_c \cdot h_c \cdot mw_c \]  

Eq. 3.182

where

- \( y_c \) is the mole fraction of component \( c \) in the gas phase, and
- \( mw_c \) is the component molecular weight.

The component gas phase enthalpy \( h_c \) has the form

\[ h_c = h_{cs} + C_{p1c} \cdot (T - T_{st}) + \frac{1}{2} \cdot C_{p2c} \cdot (T - T_{st})^2 \]

where

- \( T_{st} \) is the standard temperature defined with keyword \texttt{STCOND},
- \( h_{cs} \) is the heat of vaporization at \( T_{st} \), defined by \texttt{HEATVAPS},
- \( C_{p1c} \) is the first coefficient of the component gaseous specific heat entered with the \texttt{SPECHG} keyword, and
- \( C_{p2c} \) is the second coefficient of the component gaseous specific heat entered with this keyword.

\textbf{Note:} It is common in thermal simulation to specify the component liquid specific heats with keyword \texttt{SPECHA}. Then to obtain the enthalpy values of gaseous components, a heat of vaporization for each component is supplied with keyword \texttt{HEATVAP}. Alternatively, one can specify gas component enthalpy values with this keyword and obtain the liquid enthalpy values by again specifying the component heats of vaporization with keyword \texttt{HEATVAP}. Either \texttt{SPECHA} or \texttt{SPECHG} must be set for each run.

This keyword should only be used for thermal runs with volatile gas. For \texttt{DEADOIL} runs the hydrocarbon components do not appear in the gas phase.

The data record contains one value for each component:

1. gas hydrocarbon component specific heats, \( N_c \) values, one for each component

\textbf{UNITS:} kJ/kg/K (\texttt{METRIC}), Btu/lb/°R/°R (\texttt{FIELD}), J/gm/K (\texttt{LAB}), kJ/kg/K (\texttt{PVT-M})

\textbf{DEFAULT:} 0.0
If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of **TABDIMS**. The use of multiple EoS regions is discussed in "Equations of state regions" in the *ECLIPSE Technical Description*.

**Example**

For four components

```
SPECHH
 0.5E-4 0.5E-4 0.5E-4 0.6E-4 /
```
Solid component specific heat

This keyword can only be used with the THERMAL option.

SPECHS specifies the first coefficient of the oil component solid phase specific heats, for each EoS region.

For multi-component simulations, the solid phase molar enthalpy is the mole fraction weighted sum of the component enthalpies:

\[ h_s = \sum_{c=1}^{N_c} s_c \cdot h_c \cdot m_{w_c} \]  

Eq. 3.183

where

- \( s_c \) is the mole fraction of component \( c \) in the oil phase, and
- \( m_{w_c} \) is the component molecular weight.

The component solid enthalpy \( h_c \) has the form

\[ h_c = C_{p1c} \cdot (T - T_{st}) + \frac{1}{2} \cdot C_{p2c} \cdot (T - T_{st})^2 \]  

Eq. 3.184

where

- \( C_{p1c} \) is the first coefficient of the component solid specific heat entered with this keyword
- \( C_{p2c} \) is the second coefficient of the component solid specific heat entered with the SPECHT keyword, and
- \( T_{st} \) is the standard temperature defined with keyword STCOND.

The data record contains one value for each component:

1. Liquid hydrocarbon component specific heats, \( N_c \) values, one for each component
   
   **UNITS:** kJ/kg/K (METRIC), Btu/lb/°R (FIELD), J/gm/K (LAB), kJ/kg/K (PVT-M)
   
   **DEFAULT:** 0.0

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

**Example**

For four components

```
SPECHS
0.55 0.57 0.58 0.61 /
```
**SPECHT**

### Solid component specific heat

This keyword can only be used with the **THERMAL** option.

**SPECHT** specifies the second coefficient of the oil component solid specific heats, for each EoS region.

For multi-component simulations, the solid phase molar enthalpy is the mole fraction weighted sum of the component enthalpies:

\[
h_s = \sum_{c=1}^{N_c} x_c \cdot h_c \cdot mw_c
\]

Eq. 3.185

where

- \( x_c \) is the mole fraction of component \( c \) in the oil phase, and
- \( mw_c \) is the component molecular weight.

The component liquid enthalpy \( h_l \) has the form

\[
h_l = C_{p1c} \cdot (T - T_{st}) + \frac{1}{2} \cdot C_{p2c} \cdot (T - T_{st})^2
\]

Eq. 3.186

where

- \( C_{p1c} \) is the first coefficient of the component solid specific heat entered with the **SPECHS** keyword,
- \( C_{p2c} \) is the second coefficient of the component solid specific heat entered with this keyword, and
- \( T_{st} \) is the standard temperature defined with keyword **STCOND**.

The data record contains one value for each component:

1. liquid hydrocarbon component specific heats, \( N_c \) values, one for each component

   **UNITS**: kJ/kg/K (**METRIC**), Btu/lb/°R/°R (**FIELD**), J/gm/K (**LAB**), kJ/kg/K (**PVT-M**)

   **DEFAULT**: 0.0

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of **TABDIMS**. The use of multiple EoS regions is discussed in "Equations of state regions" in the **ECLIPSE Technical Description**.

### Example

For four components

```plaintext
SPECHT
0.005 0.007 0.008 0.006 /
```

---

**Keywords**

2138
**SPECHW1 Water component specific heat**

This keyword can only be used with the **THERMAL** option.

SPECHW1 specifies the first coefficient of the water component liquid specific heats, for each EoS region.

For multi-component simulations, the water phase molar enthalpy is the mole fraction weighted sum of the component enthalpies:

\[
h_w = \sum_{c=1}^{N_c} a_c \cdot h_c \cdot m_w c
\]

where

- \(a_c\) is the mole fraction of component \(c\) in the aqua phase, and
- \(m_w c\) is the component molecular weight.

The component liquid enthalpy \(h_c\) has the form

\[
h_c = C_{p1c} \cdot (T-T_{st}) + \frac{1}{2} \cdot C_{p2c} \cdot (T-T_{st})^2
\]

where

- \(C_{p1c}\) is the first coefficient of the component liquid specific heat entered with this keyword
- \(C_{p2c}\) is the second coefficient of the component liquid specific heat entered with the **SPECHW2** keyword, and
- \(T_{st}\) is the standard temperature defined with keyword **STCOND** and \(T\) the temperature.

*Note:* The \(\text{H}_2\text{O}\) component enthalpy is computed internally. In a **CO2STORE** run with the \(\text{H}_2\text{O}\) component specified in **CNAMES**, defaulting the value for \(\text{H}_2\text{O}\) means that the enthalpy is computed similarly to a standard case.

This keyword should only be used for thermal runs where the hydrocarbons dissolve in the water phase, see keyword **KW1TABTn**, or in thermal **CO2STORE** runs.

The data record contains one value for each component:

1. Aqua hydrocarbon component specific heats, \(N_c\) values, one for each component.
   
   **UNITS:** kJ/kg/K (**METRIC**), Btu/lb/°R (**FIELD**), J/gm/K (**LAB**), kJ/kg/K (**PVT-M**)
   
   **DEFAULT:** 0.0

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of **TABDIMS**. The use of multiple EoS regions is discussed in "Equations of state regions" in the **ECLIPSE Technical Description**.

---

**Keywords**

2139
Example

For four components

```
SPECMW1
0.55 0.57 0.58 0.61 /
```
This keyword can only be used with the THERMAL option.

*SPECHW2* specifies the second coefficient of the aqua component liquid specific heats, for each EoS region.

For multi-component simulations, the aqua phase molar enthalpy is the mole fraction weighted sum of the component enthalpies:

\[
h_w = \sum_{c=1}^{N_c} a_c \cdot h_c \cdot mw_c
\]

Eq. 3.189

where

\(a_c\) is the mole fraction of component \(c\) in the aqua phase, and

\(mw_c\) is the component molecular weight.

The component liquid enthalpy \(h_c\) has the form

\[
h_c = C_{p1c} \cdot (T-T_{st}) + \frac{1}{2} \cdot C_{p2c} \cdot (T-T_{st})^2
\]

Eq. 3.190

where

\(C_{p1c}\) is the first coefficient of the component liquid specific heat entered with the *SPECHW1* keyword,

\(C_{p2c}\) is the second coefficient of the component liquid specific heat entered with this keyword, and

\(T_{st}\) is the standard temperature defined with keyword *STCOND* and \(T\) the temperature.

*Note:* The H\(_2\)O component enthalpy is computed internally. In a CO2STORE run with the H\(_2\)O component specified in CNUMS, defaulting the value for H\(_2\)O means that the enthalpy is computed similarly to a standard case.

The data record contains one value for each component:

1. Liquid hydrocarbon component specific heats, \(N_c\) values, one for each component.

   - **UNITS:** kJ/kg/K (METRIC), Btu/lb/°R (FIELD), J/gm/K (LAB), kJ/kg/K (PVT-M)
   - **DEFAULT:** 0.0

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

**Example**

For four components
| SPECHW1 | 0.005 0.007 0.008 0.006 / |
**SPECROCK  Rock specific heat data**

This keyword should only be used for runs with the Temperature option active (see keyword TEMP).

The data comprises NTSFUN (see keyword TABDIMS) tables, defining the volume specific heat of rock as a function of temperature. Each table consists of 2 columns of data and is terminated by a slash (/):

1. The temperature.
   
   The values should increase monotonically down the column.
   
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

2. The corresponding volume specific heat of rock.
   
   UNITS: kJ/m³/K (METRIC), Btu/ft³/°R (FIELD), J/cc/K (LAB), kJ/m³/K (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see keyword TABDIMS).

See also the SPECHEAT keyword and the "Temperature option" in the ECLIPSE Technical Description.

**Example**

With NTSFUN=2 and NSSFUN ≥2:

```
SPECROCK
  0.0  25.0
  300.0  25.0
/
  0.0  27.0
  300.0  27.0
/
```
Initial polymer concentrations

This keyword sets the initial polymer concentration in runs where the initial conditions are specified by enumeration.

The keyword should be followed by one real number for every grid block specifying the initial polymer concentration. The data field should be terminated by a slash (/).

UNITs: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scc (LAB)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to $N_{DIVIX}$), followed by the Y axis index (from 1 to $N_{DIVIY}$), and finally the Z axis index (from 1 to $N_{DIVIZ}$). $N_{DIVIX}$, $N_{DIVIY}$ and $N_{DIVIZ}$ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*0.18). Note that spaces must not be inserted on either side of the asterisk (*).

See also the keywords PRESSURE, SGAS, RS, PBUB, RV, PDEW and SWAT in the SOLUTION section. Refer to "Polymer flood model" in the ECLIPSE Technical Description for further information.

Note: When adsorption is active, the concentration specified here will be honored and extra adsorbed polymer is added to the rock so that it is in equilibrium with the input fluid concentration.

Note: If a non-zero concentration is specified in cells containing none of the host phase, the concentration will be automatically reset to zero. This behavior can be turned off using item 165 in OPTIONS.

Example

With $N_{DIVIX}$=10, $N_{DIVIY}$=1 and $N_{DIVIZ}$=2 specified in RUNSPEC keyword DIMENS:

```
SPOLY
10*0.0
9*0.0 25.0 /
```
Reference pressures for solid phase

In a run with \( N_c \) components, this keyword specifies the reference pressures at which the hydrocarbon solid densities are defined. This keyword can only be used in runs with solids (keyword \texttt{SOLID} in the \texttt{RUNSPEC} section).

The expression used to calculate component solid densities is the following:

\[
\rho_c = \rho_{\text{ref},c} \left[ 1 - c_{P_c} \left( P_c - P_{\text{ref},c} \right) \right] \left[ 1 + c_{T_1} c_{P_c} \left( T - T_{\text{ref},c} \right) \right]
\]

where

- \( \rho_{\text{ref},c} \) is the reference density, specified with keyword \texttt{SDREF}
- \( c_{P_c} \) is the component solid compressibility, specified with keyword \texttt{SCREF}
- \( P_{\text{ref},c} \) is the reference pressure, specified with this keyword
- \( T_{\text{ref},c} \) is the reference temperature, specified with keyword \texttt{STREF}
- \( c_{T_1} \) is the thermal expansion coefficient, specified with keyword \texttt{STHERMX1}.

There should be one value for each component:

- UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
- DEFAULT: the standard pressure set with keyword \texttt{STCOND}.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword \texttt{TABDIMS}. The use of multiple EoS regions is discussed in “Equations of state regions” in the \textit{ECLIPSE Technical Description}.

Example

```
SPREF
1527.4
1527.4 /
```

\textbf{Keywords}

2145
**Gas/solvent saturation functions**

This keyword may be used with the solvent model as described in "Solvent model" in the ECLIPSE Technical Description.

The data comprises NTSFUN (see keyword TABDIMS) tables of gas/solvent saturation functions, each terminated by a slash (/).

The tables consist of 3 columns of data. The first column is the phase fraction of either the gas or the solvent \((S_{gas} / (S_{gas} + S_{solvent})\) or \((S_{solvent} / (S_{gas} + S_{solvent})\)). The next two columns are respectively the relative permeability multiplier for the gas and the solvent \((K^*_{rg} \text{ and } K^*_s)\).

The relative permeabilities of the 2 components are given by:

\[
K^*_{rg} = K_{rgt} (S_{gas} + S_{solvent}) K^*_{rg} \\
K^*_s = K_{rgt} (S_{gas} + S_{solvent}) K^*_s
\]

where \(K_{rgt}\) is the total gas phase relative permeability input using the SGFN keyword.

Column:

1. The phase fraction.
   
   Values should span the range 0 to 1 and increase monotonically down the column.

2. The corresponding gas relative permeability multiplier.
   
   Values should be between 0 and 1 and should be level or increasing down the column.

3. The corresponding solvent relative permeability multiplier.
   
   Values should be between 0 and 1 and should be level or increasing down the column.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see keyword TABDIMS).

**Example**

With NTSFUN=2 and \(\text{NSSFUN} \geq 6\)

<table>
<thead>
<tr>
<th>SSFN</th>
<th>Krg*</th>
<th>Krs*</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>/</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.2</td>
<td>0.1</td>
<td>0.2</td>
</tr>
<tr>
<td>0.4</td>
<td>0.2</td>
<td>0.4</td>
</tr>
<tr>
<td>0.6</td>
<td>0.4</td>
<td>0.5</td>
</tr>
<tr>
<td>0.8</td>
<td>0.7</td>
<td>0.8</td>
</tr>
<tr>
<td>1.0</td>
<td>1.0</td>
<td>1.0</td>
</tr>
</tbody>
</table>

Keywords

2146
**SSHIFT**

**Equation of state shift parameters**

In a run with \( N_c \) components, this keyword specifies volume shift parameters in dimensionless form. The use of these shift parameters is described in "Equations of state" in the *ECLIPSE Technical Description*. This shift parameter can be applied to any equation of state.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword `TABDIMS`. The use of multiple EoS regions is discussed in "Equations of state regions" in the *ECLIPSE Technical Description*.

**Examples**

**Example 1**

```plaintext
-- 3-Parameter EoS Shift Coefficients
SSHIFT
-0.07361 -0.15427 -0.05559 -0.02573 -0.08493 -0.16602 /
```

**Example 2**

```plaintext
-- With two reservoir EoS regions
RTEMP
212.0 /
223.0 /
```
SSHIFTS  

Equation of state shift parameters for surface EoS

In a run with $N_c$ components, this keyword specifies volume shift parameters in dimensionless form. The use of these shift parameters is described in "Equations of state" in the ECLIPSE Technical Description.

This shift parameter can be applied to any equation of state.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

Example

```
-- 3-Parameter EoS Shift Coefficients for surface facilities
SSHIFTS
-0.07361 -0.15427 -0.05559 -0.02573 -0.08493 -0.16602 /
```
**Initial solvent saturations**

This keyword sets the initial solvent saturation in runs where the initial conditions are specified by enumeration.

The keyword should be followed by one real number for every grid block specifying the initial solvent saturation. The data field should be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest (from 1 to \( \text{NDIVIX} \)), followed by the Y axis index (from 1 to \( \text{NDIVIY} \)), and finally the Z axis index (from 1 to \( \text{NDIVIZ} \)). \( \text{NDIVIX} \), \( \text{NDIVIY} \) and \( \text{NDIVIZ} \) are specified in RUNSPEC keyword **DIMENS**. Repeat counts may be used for repeated values (for example 115*0.18). Note that spaces must not be inserted on either side of the asterisk (*).

See also the keywords **PRESSURE**, **SGAS**, **RS**, **PBUB**, **RV**, **PDEW** and **SWAT** in the **SOLUTION** section.

Refer to "Solvent model" in the *ECLIPSE Technical Description* for more information.

**Example**

With \( \text{NDIVIX}=10 \), \( \text{NDIVIY}=1 \) and \( \text{NDIVIZ}=2 \) specified in RUNSPEC keyword **DIMENS**:

```
SSOL
10*0.0
9*0.0 0.5 /
```
SSOLID

Specifies cell initial solid saturation values

This keyword specifies the initial solid saturation values for each cell. It can only be used with the SOLID option. If this keyword is used then keyword SMF should also be used to define the solid composition.

The keyword should be followed by one real number for each grid block in the current input box, terminated with a slash (/).

Example

```
SSOLID
4000*0.01 /
```
START

Specifies a start date

This keyword specifies the start date of the simulation. (Any report dates entered in the DATES keyword in the SCHEDULE section must be later than the start date.)

The keyword is followed by the following items of data, denoting the day, month, year and time of the start of the simulation, terminated with a slash (/).

1. Day
   Day of the month (an integer between 1 and 31).

2. Month
   Name of the month (JAN, FEB, MAR, APR, MAY, JUN, JUL, AUG, SEP, OCT, NOV, DEC).
   For July, the abbreviation JLY is a valid alternative.

3. Year
   The year (a four-figure integer).

4. Time
   The time (24 hour, in the format HH:MM:SS.SSSS)

**ECLIPSE 100**
DEFAULT: 1 Jan 1983 00:00:00

**ECLIPSE 300**
DEFAULT: 1 Jan 1990 00:00:00

Example

```
START
12 FEB 1987 /
```
STCOND

Specify standard conditions

A standard temperature and pressure must be specified.

This used to obtain surface volumes at stock tank conditions. The keyword should be followed by two data items terminated with a slash (/):

1. Standard temperature
   
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
   
   DEFAULT: 15.56°C (METRIC, LAB and PVT-M), 60°F (FIELD)

2. Standard pressure
   
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
   
   DEFAULT: 1.01325 barsa (METRIC), 14.6959 psia (FIELD), 1 atma (LAB and PVT-M)

Related keywords

For well production, surface volumes are calculated using a well separator, defined by the WSEPCOND keyword. If a well separator is not defined, the group separator defined by the SEPCOND keyword is used. If the group separator is not defined, the field separator defined by the FIELDSEP keyword is used. If the field separator is not defined, a single flash to standard conditions given by this keyword STCOND is used.

For fluid-in-place calculations, surface volumes are calculated using region separators, defined by the FIPSEP keyword. If the region separator is not defined, the field separator FIELDSEP is used. If the field separator is not defined, a single flash to standard conditions given by this keyword STCOND is used.

Example

```
STCOND
60 14.65 /
```
STEST

Instructions for periodic testing of closed secondary wells

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Secondary well name or well name template

A template enclosed in quotes can be used to refer to multiple wells. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. The secondary well name must have been previously associated with a well segment of an existing multisegment well using the WSEGWELL keyword.

2. Testing interval

This must be greater than zero.

UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

The secondary well is tested at the beginning of the first timestep that starts after the specified interval has elapsed since it was closed. Subsequent tests are performed at each timestep that starts after the specified interval has elapsed since the previous test.

If the testing interval is very small (for example, 1 day), the well is tested at the beginning of each timestep (unless of course the timesteps are smaller than this).

The test is applied to secondary wells which have been shut because the segment pressure has reached the injection or production limit or because the well to which the segment belongs has been shut.

End the data record with a slash (/).

End the set of records with a slash (/).

Example

```
STEST
  'PROD-S*' 5 /
/
```
STHERMX1  Thermal expansion coefficient for solid phase

This keyword specifies the first thermal expansion coefficient for each of the \( N_c \) hydrocarbon components in the solid phase. It can only be used in runs with solids (keyword `SOLID` in the `RUNSPEC` section).

The expression used to calculate component solid densities is the following:

\[
\rho_c = \frac{\rho_{ref,c}}{\left[ 1 + c_{P_c} \left( P - P_{ref,c} \right) \right] \left[ 1 + c_{T_{1,c}} \left( T - T_{ref,c} \right) \right]}
\]

Eq. 3.193

where

- \( \rho_{ref,c} \) is the reference density, specified with keyword `SDREF`,
- \( c_{P_c} \) is the component solid compressibility, specified with keyword `SCREF`,
- \( P_{ref,c} \) is the reference pressure, specified with keyword `SPREF`,
- \( T_{ref,c} \) is the reference temperature, specified with keyword `STREF`,
- \( c_{T_{1,c}} \) is the thermal expansion coefficient, specified with this keyword.

There should be one value for each component:

- UNITS: 1/K (METRIC), 1/°R (FIELD), 1/K (LAB), 1/K (PVT-M)
- DEFAULT: 0.0

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of `TABDIMS`. The use of multiple EoS regions is discussed in "Equations of state regions" in the `ECLIPSE Technical Description`.

Example

```
STHERMX1
-- thermal expansion coefficients for a system with 3 hydrocarbon
-- components
 1* 3.5E-06 1* /
```
Oil-gas surface tension versus pressure

This keyword supplies tables of oil-gas surface tension versus pressure. They are used by two facilities in ECLIPSE: the SURFTENS option and multisegment wells. The keyword should not be present if neither of these options are in use.

The SURFTENS option calculates a pressure-dependent adjustment to the capillary pressure. It is activated by requesting the SURFTENS option in the RUNSPEC section keyword SATOPTS. If the STOG keyword is not supplied, the oil-gas capillary pressure retains the values specified in the keyword SGFN, SGOF or SLGOF, independently of the pressure.

The keyword may also be used if there are any multisegment wells in the run. The phase interfacial tensions are required by the drift flux model (see "The drift flux slip model" in the ECLIPSE Technical Description). If surface tension data is supplied for the table corresponding to the well’s PVT region number (see WELSPECS keyword item 11), the interfacial tension is taken from this table. If there is no surface tension data supplied for the well’s PVT table number, the interfacial tension is calculated from a built-in correlation.

The data comprises NTPVT (see keyword TABDIMS in the RUNSPEC section) tables of oil-gas surface tension versus pressure.

Each table consists of two records, each record terminated with a slash (/).

**Record 1**

A reference oil phase pressure, at which the oil-gas capillary pressure has the values specified in keyword SGFN, SGOF or SLGOF.

(This data item is only used by the SURFTENS option. It is not needed to calculate the interfacial tension in the multisegment well drift flux model, and the value entered here does not affect its results.)

UNITs: barsa (METRIC), psia (FIELD), atma (LAB)

**Record 2**

A table consisting of two columns of data.

1. The oil phase pressure. Values should increase monotonically down the column.
   
   UNITs: barsa (METRIC), psia (FIELD), atma (LAB)

2. The corresponding oil-gas surface tension.
   
   UNITs: N/m (METRIC), lbf/in (FIELD), dynes/cm (LAB)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see keyword TABDIMS in the RUNSPEC section).

**Calculations for the SURFTENS option**

The oil-gas capillary pressures (entered in keyword SGFN, SGOF or SLGOF) are adjusted according to the ratio of the surface tension at the grid block oil phase pressure to the surface tension at the reference pressure. The capillary pressures input in the SGFN, SGOF or SLGOF keyword are assumed to be at the reference pressure specified in the STOG keyword.

Thus the effective oil-gas capillary pressure is:
\[ P_{\text{cog}} \text{ (oil pressure)} = P_{\text{cog}} \text{ (input by user)} \cdot \frac{\text{ST (oil pressure)}}{\text{ST (reference pressure)}} \]  

**Eq. 3.194**

**Note:** The surface tension is only used as a ratio in the **SURFTENS** option, and hence the units used does not affect the results of the calculation.

**Example**

With **NTPVT=1** and **NPPVT \geq 11**:

```
SURFTENS

-- PRESSURE   SURFACE TENSION
  1674      6.0
  2031      4.7
  2530      3.3
  2991      2.2
  3553      1.28
  4110      0.72
  4544      0.444
  4935      0.255
  5255      0.155
  5545      0.090
  7000      0.050
```

**Keywords**

2156
STONE Requests Stone II oil relative permeabilities

The default three-phase relative permeability is the linear combination described in "Saturation functions" in the ECLIPSE Technical Description. As an alternative, this keyword requests the use of the Stone’s method II.

The STONE keyword contains no items.

STONE is an alias for STONE2.
STONEx

Request Stone three-phase oil relative permeability model

This keyword, which has no associated data, is used to specify that the three-phase oil relative permeability values are to be calculated using the modified Stone’s formula number 1. The STONE1 keyword should only be used in three-phase runs.

ECLIPSE uses the default three-phase oil relative permeability model if the keywords STONE1, STONE2, STONE, IKU3P or ODD3P are omitted from the PROPS section of the input data file. Tabulated output of the three-phase oil relative permeability values may be obtained using the mnemonic SOF2 or SOF3 in the RPTPROPS keyword.

The minimum oil saturation used by the STONE1 model can optionally be input as a table of minimum oil saturation versus either water or gas saturation, by using the SOMWAT or SOMGAS keyword. If neither of these keywords are present, the minimum oil saturation is taken as MIN(S_{owcr}, S_{ogcr}), the minimum of the critical oil-to-water saturation and the critical oil-to-gas saturation.

See "Saturation functions" in the ECLIPSE Technical Description for a description of three-phase oil relative permeability models available in ECLIPSE.

Example

```
STONE1
```

Keywords

2158
STONE1EX  Stone’s first three-phase oil relative permeability model exponent term

This keyword specifies the value of the exponent which is to be applied to the combination of saturation terms which are used in the calculation of the oil relative permeability when Stone’s first three phase relative permeability model has been specified. The STONE1EX keyword should only be used in three-phase runs for which STONE1 has also been specified.

The data associated with this keyword consists of one exponent value per saturation table. The number of saturation tables is specified in item 1 (NTSFUN) of keyword TABDIMS in the RUNSPEC section. If hysteresis is being modeled, which is defined by the SATOPTS keyword in the RUNSPEC section, the exponent values assigned to the drainage and imbibition tables should be equal for each saturation region.

The value of the exponent is constrained to lie between a minimum value of 0.01 and a maximum value of 100.0. If the value for the first table is defaulted, the exponent will be assigned a value of 1.0. Where values are defaulted for the second and successive tables, these will be assigned to the value of the exponent for the previous table.

See "Saturation functions" in the ECLIPSE Technical Description for a description of three-phase oil relative permeability models available in ECLIPSE.

Example

With NTSFUN =3:

```
STONE1EX
  1.0 / 
  4.0 / 
  1.0 / 
```

Keywords

2159
STONET2

Request Stone three-phase oil relative permeability model

This keyword, which has no associated data, is used to specify that the three-phase oil relative permeability values are to be calculated using the modified form of Stone’s model number 2. The STONE2 keyword should only be used in three-phase runs.

ECLIPSE uses the default three-phase oil relative permeability model if the keywords STONE1, STONE2, STONE, IKU3P or ODD3P are omitted from the PROPS section of the input data file. Tabulated output of the three-phase oil relative permeability values may be obtained using the mnemonic SOF2 or SOF3 in the RPTPROPS keyword.

See "Saturation functions" in the ECLIPSE Technical Description for a fuller description of the three-phase oil relative permeability models available in ECLIPSE.

Note that any negative oil relative permeabilities produced by Stone’s model 2 are automatically changed to zero by ECLIPSE.

Example

```
STONE2
```

Keywords

2160
**STONEPAR**

**Modify the Stone I three-phase oil relative permeability**

This keyword sets two coefficients, A and B, which define a modification of the Stone I three-phase relative permeability (see the STONE1 keyword). These modify the mobile oil value, \( S_{om} \). This is described in more detail in "Saturation functions" in the ECLIPSE Technical Description.

The STONEPAR keyword allows you to make the mobile oil value \( S_{om} \) a function of the gas saturation. In the absence of STONEPAR this is fixed as the minimum of the critical oil saturations in water or gas, \( \text{MIN}(S_{owcr}, S_{ogcr}) \). STONEPAR makes this a function of \( S_g \) as:

\[
S_{om} = S_{ogcr}X_g^A + S_{owcr}(1-X_g)^A + B S_{gm}^2 X_g(1-X_g)
\]

where:

\[
S_{gm} = 1-S_{wco} - S_{ogcr}
\]

\[
X_g = \frac{S_g}{S_{gm}}
\]

Eq. 3.195

This expression interpolates \( S_{om} \) between \( S_{ogcr} \) when the gas saturation is at its maximum value and \( S_{owcr} \) when the gas saturation is at its minimum value - so that \( S_{om} \) reduces to its correct two-phase limits. Values of \( A=1 \) and \( B=0 \) would yield linear interpolation between these limits.

STONEPAR is followed by the two parameters, followed by a slash (/).

1. Interpolation parameter A
   
   DEFAULT: 1.0

2. Interpolation parameter B
   
   DEFAULT: 0.0
Figure 3.16. Linear interpolation for Som between Sogcr and Sowcr

The B value enables curvature to be added to the interpolation. A positive value of B would yield an interpolant of the form:
Figure 3.17. Curved interpolation for Som between Sogcr and Sowcr

The default values of A and B are 1 and 0 respectively. Great care should be taken if negative values of B are used to avoid negative values.

Note that using STONEPAR with default values is not the same as omitting the keyword.

Example

```
STONEPAR
   --Interpolate Som with positive curvature.
   1.0 0.05 /
```
This keyword defines the stoichiometric coefficients for the products in each chemical reaction.

The keyword is followed by one record for each reaction defined by the `REACTION` keyword. Each record contains $N_c$ items, one for each component, $N_c + 1$ items if water is included, or $N_c + N_w$ if the multi-component water model is included (for example, in the case of surfactant adsorption or desorption).

DEFAULT: 0

Items should be defaulted or set to zero for components that are not products. Otherwise they should be set to the stoichiometric coefficient of the component in the corresponding reaction.

See "Chemical reactions" in the *ECLIPSE Technical Description*.

**Example**

For a run with four components $C_{12}H_{26}$, $C_3H_8$, $O_2$, $CO_2$ and water, the two reactions

$$C_{12}H_{26} + 18.5 \cdot O_2 \rightarrow 12 \cdot CO_2 + 13 \cdot H_2O$$

$$C_3H_8 + 5 \cdot O_2 \rightarrow 3 \cdot CO_2 + 4 \cdot H_2O$$

can be defined using the `STOREAC` and `STOPROD` keywords:

```plaintext
STOREAC
---HEAVY LIGHT O2 CO2 H2O
 1 0 18.5 0 0 /
 0 1 5 0 0 /

STOPROD
---HEAVY LIGHT O2 CO2 H2O
 0 0 0 12 13 /
 0 0 0 3 4 /
```
**STOREAC**

**Stoichiometric coefficients for reactants**

This keyword defines the stoichiometric coefficients for the reactants in each chemical reaction.

The keyword is followed by one record for each reaction defined by the `REACTION` keyword. Each record contains \( N_c \) items, one for each component, \( N_c + 1 \) items if water is included, or \( N_c + N_w \) if the multi-component water model is included (for example, in the case of surfactant adsorption or desorption).

DEFAULT: 0

Items should be defaulted or set to zero for components that are not reactants. Otherwise they should be set to the stoichiometric coefficient of the component in the corresponding reaction.

See "Chemical reactions" in the *ECLIPSE Technical Description*.

**Example**

For a run with four components \( \text{C}_{12}\text{H}_{26} \), \( \text{C}_{3}\text{H}_{8} \), \( \text{O}_2 \), \( \text{CO}_2 \) and water, the two reactions

\[
\text{C}_{12}\text{H}_{26} + 18.5 \cdot \text{O}_2 \rightarrow 12 \cdot \text{CO}_2 + 13 \cdot \text{H}_2\text{O}
\]

\[
\text{C}_{3}\text{H}_{8} + 5 \cdot \text{O}_2 \rightarrow 3 \cdot \text{CO}_2 + 4 \cdot \text{H}_2\text{O}
\]

can be defined using the `STOREAC` and `STOPROD` keywords:

```
STOREAC
--HEAVY  LIGHT  O2   CO2  H2O
 1      0      18.5  0    0  /
 0      1      5     0    0  /

STOPROD
--HEAVY  LIGHT  O2   CO2  H2O
 0      0      0     12   13 /
 0      0      0     3    4  /
```
Oil-water surface tension versus pressure

This keyword supplies tables of oil-water surface tension versus pressure. They are used by two facilities in ECLIPSE: the SURFTENS option and multisegment wells. The keyword should not be present if neither of these options are in use.

The SURFTENS option calculates a pressure-dependent adjustment to the capillary pressure. It is activated by requesting the SURFTENS option in the RUNSPEC section keyword SATOPTS. If the STOW keyword is not supplied, the oil-water capillary pressure retains the values specified in the keyword SWFN or SWOF, independently of the pressure.

The keyword may also be used if there are any multisegment wells in the run. The phase interfacial tensions are required by the Drift Flux model (see "The drift flux slip model" in the ECLIPSE Technical Description). If surface tension data is supplied for the table corresponding to the well’s PVT region number (see WELSPECS keyword item 11), the interfacial tension is taken from this table. If there is no surface tension data supplied for the well’s PVT table number, the interfacial tension will be calculated from a built-in correlation.

The data comprises NTPVT (see keyword TABDIMS in the RUNSPEC section) tables of oil-water surface tension versus pressure.

Each table consists of two records, each record terminated with a slash (/).

**Record 1**

A reference oil phase pressure, at which the oil-water capillary pressure has the values specified in keyword SWFN or SWOF.

(This data item is only used by the SURFTENS option. It is not needed to calculate the interfacial tension in the multisegment well drift flux model, and the value entered here does not affect its results.)

UNITs: barsa (METRIC), psia (FIELD), atma (LAB)

**Record 2**

A table consisting of two columns of data.

1. The oil phase pressure. Values should increase monotonically down the column.
   
   UNITs: barsa (METRIC), psia (FIELD), atma (LAB)

2. The corresponding oil-water surface tension.
   
   UNITs: N/m (METRIC), lbf/in (FIELD), dynes/cm (LAB)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see keyword TABDIMS in the RUNSPEC section).

**Calculations for the SURFTENS option**

The oil-water capillary pressures (input in keyword SWFN or SWOF) are adjusted according to the ratio of the surface tension at the grid block oil phase pressure to the surface tension at the reference pressure. The capillary pressures input in the SWFN or SWOF keyword are assumed to be at the reference pressure specified in the STOW keyword.

Thus the effective oil-water capillary pressure is:
\[ P_{cow} \text{ (oil pressure)} = P_{cow} \text{ (input by user)} \frac{ST \text{ (oil pressure)}}{ST \text{ (reference pressure)}} \quad \text{Eq. 3.198} \]

**Note:** The surface tension is only used as a ratio in the SURFTENS option, and hence the units used does not affect the results of the calculation.

**Example**

With NTPVT=1 and NPPVT ≥ 6:

```
STOW
-- REFERENCE PRESSURE
-- 5500 /
-- PRESSURE SURFACE TENSION
-- 1000  10.0
   2000  9.8
   3000  8.6
   4000  7.4
   5000  5.3
   6000  1.2 /
```
Reference temperatures for solid phase

In a run with $N_c$ components this keyword specifies the reference temperatures at which solid densities are defined. The keyword can be used in runs with solids (keyword `SOLID` in the RUNSPEC section).

In a live-oil thermal run, the component solid densities are given by:

$$
\rho_c = \frac{\rho_{ref,c}}{\left[1 + c_{P_c}(P - P_{ref,c})\right]\left[1 + c_{T1_c}(T - T_{ref,c})\right]}
$$

where

- $\rho_{ref,c}$ is the reference density, specified with keyword `SDREF`,
- $c_{P_c}$ is the component solid compressibility, specified with keyword `SCREF`,
- $P_{ref,c}$ is the reference pressure, specified with keyword `SPREF`,
- $T_{ref,c}$ is the reference temperature, specified with this keyword,
- $c_{T1_c}$ is the thermal expansion coefficient, specified with keyword `STHERMX1`.

There should be one value for each component:

**UNITS:** K (METRIC), °R (FIELD), K (LAB), K (PVT-M)

**DEFAULT:** the standard temperature set with keyword `STCOND`

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword `TABDIMS`. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

**Example**

For a two component run, with one EoS region:

```plaintext
STREF
527.4 527.4 /
```
STRESBC  Stress boundary conditions for coupled stress and fluid flow

This keyword is being replaced by new keywords for setting traction, stress and displacement boundary conditions. Please refer to keywords GMTRABC, GMPSTBC and GMDISBC.

This keyword allows either traction or displacement boundary conditions to be specified on all boundaries. A traction at a boundary surface is a force/unit area. If a particular boundary is not specified in this keyword, then that boundary is defaulted to a rigid boundary.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the traction or velocity boundary condition is applied
   
   This must be one of the following:
   
   I+  Specify the boundary condition on the outer I+ edge
   I-  Specify the boundary condition on the outer I- edge
   J+  Specify the boundary condition on the outer J+ edge
   J-  Specify the boundary condition on the outer J- edge
   K+  Specify the boundary condition on the outer K+ edge
   K-  Specify the boundary condition on the outer K- edge

   DEFAULT: There is no default for this item; it must be specified.
   
   The K- boundary is the top boundary of the simulation, K+ is the bottom.

2. An integer specifying the type of boundary condition on this face
   
   This must be one of the following:
   
   1 A total normal traction is specified on the boundary denoted in item 1. The magnitude of this normal traction is set in item 3. The sign of the traction is +ve for tension, -ve for compression. (See also item 6 of the GEOMECH keyword.)

   2 A normal rock displacement velocity is specified on the boundary denoted in item 1.

   3 A given traction with X, Y, Z components. A single X, Y, Z traction component is set in item 3 and the component name is set in item 13. Separate lines are needed to set different components. As for Type 1, the sign of the stress is +ve for tension, -ve for compression. (See also item 6 of the GEOMECH keyword.)
4. A rock displacement velocity with X, Y, Z components. A single X, Y, Z velocity component is set in item 6 and the component name must be set in item 13. Separate lines are needed to set different components.

For boundary condition types 3 and 4, the X, Y, Z components conform to the X, Y, Z directions as specified in the COORD keyword for corner point geometry.

DEFAULT: 2

3. A normal traction on this boundary, if the boundary condition type in item 2 is 1
X or Y or Z component of a given traction, if the boundary condition type in item 2 is 3. The component direction is set in item 13 below. Normal component or X component if type in item 2 is 3.

The sign of the traction is +ve for tension, -ve for compression. See the example below. (See also item 6 of the GEOMECH keyword.)

UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

DEFAULT: 0

4. Y component if type in item 2 is 3
UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

5. Z component if type in item 2 is 3
UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

6. A normal rock velocity on this boundary, if the boundary condition type in item 2 is 2
X or Y or Z component of a rock velocity vector, if the boundary condition type in item 2 is 4.
This is typically 0 to specify a rigid boundary.

Velocity vector orientation follows the orientation of the coordinate system.

UNITS: m/day (METRIC), ft/day (FIELD), cm/hr (LAB), m/day (PVT-M)

DEFAULT: 0

7. First block to be modified in the I direction
DEFAULT: 1 if an I- face

8. Last block to be modified in the I direction
DEFAULT: NX if an I+ face

9. First block to be modified in the J direction
DEFAULT: 1 if a J- face

10. Last block to be modified in the J direction
DEFAULT: NY if a J+ face

11. First block to be modified in the K direction
DEFAULT: 1 if a K- face

12. Last block to be modified in the K direction
DEFUALT: NZ if a K+ face

13. A character string that gives the component of traction or velocity if the boundary type in item 2 is either 3 or 4. This must be one of ‘X’, ‘Y’ or ‘Z’ and is required for type 3 of type 4 boundaries.

DEFUALT: None. It must be specified if item 2 is either 3 or 4

**Examples**

**Example 1**

--- set a 3000 psi total compressive traction on the x+ and y+ boundaries
--- set a 3000 psi overburden pressure (compressive)
--- note that the ‘K-’ direction is positive downwards

```
STRESBC
-- side type total traction
 'I+'   1   -3000 /
 'J+'   1   -3000 /
 'K-'   1   -3000 /
```

**Example 2**

--- Finite Element:
--- Velocities in X dir on I- and I+ boundaries are zero
--- Displacements in Y and Z dirs on I- and I+ boundaries are free
--- Velocities in Y dir on J- and J+ boundaries are zero
--- Displacements in X and Z dirs on J- and J+ boundaries are free
--- Velocities in Z dir on K- and K+ boundaries are zero
--- Displacements in X and Y dirs on K- and K+ boundaries are free

```
STRESBC
-- side type SXYZ SHEARS VXYZ range comp
 'I-'   4   3*          0    6*   'X' /
 'I-'   3   0    9*               'Y' /
 'I-'   3   0    9*               'Z' /
 'I+'   4   3*          0    6*   'X' /
 'I+'   3   0    9*               'Y' /
 'I+'   3   0    9*               'Z' /
 'J-'   4   3*          0    6*   'Y' /
 'J-'   3   0    9*               'Z' /
 'J-'   3   0    9*               'X' /
 'J+'   4   3*          0    6*   'Y' /
 'J+'   3   0    9*               'Z' /
 'J+'   3   0    9*               'X' /
 'K-'   4   3*          0    6*   'Z' /
 'K-'   3   0    9*               'X' /
 'K-'   3   0    9*               'Y' /
 'K+'   4   3*          0    6*   'Z' /
 'K+'   3   0    9*               'X' /
 'K+'   3   0    9*               'Y' /
```

**Keywords**

ECLIPSE Reference Manual

```
<table>
<thead>
<tr>
<th>Keywords</th>
</tr>
</thead>
<tbody>
<tr>
<td>2171</td>
</tr>
</tbody>
</table>
```
STRESBCL  Stress boundary conditions for coupled stress and fluid flow with an LGR

This keyword is being replaced by new keywords for setting traction, stress and displacement boundary conditions. Please refer to keywords GMTRABCL, GMPSTBCL, and GMDISBCL.

This keyword specifies boundary conditions when a coupled rock stress and fluid flow calculation is required. It is only applicable when the keyword GEOMECH has been specified in the RUNSPEC section.

This keyword allows either traction or displacement boundary conditions to be specified on all boundaries in a local grid refinement. A traction at a boundary surface is a force/unit area. If a particular boundary is not specified in this keyword, then that boundary is defaulted to a rigid boundary.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the traction or velocity boundary condition is applied
   This must be one of the following:
   I+  Specify the boundary condition on the outer I+ edge
   I-  Specify the boundary condition on the outer I- edge
   J+  Specify the boundary condition on the outer J+ edge
   J-  Specify the boundary condition on the outer J- edge
   K+  Specify the boundary condition on the outer K+ edge
   K-  Specify the boundary condition on the outer K- edge

   DEFAULT: There is no default for this item; it must be specified.
   The K- boundary is the top boundary of the simulation, K+ is the bottom.

2. An integer specifying the type of boundary condition on this face
   This must be one of the following:
   1 A total normal traction is specified on the boundary denoted in item 1. The magnitude of this normal traction is set in item 3. The sign of the traction is +ve for tension, -ve for compression. (See also item 6 of the GEOMECH keyword.)
   2 A normal rock displacement velocity is specified on the boundary denoted in item 1.
   3 A given traction with X, Y, Z components. A single X, Y, Z traction component is set in item 3 and the component name is set in item 13. Separate lines are needed to set different components.
As for Type 1, the sign of the stress is +ve for tension, -ve for compression. (See also item 6 of the GEOMECH keyword.)

4 A rock displacement velocity with X, Y, Z components. A single X, Y, Z velocity component is set in item 6 and the component name must be set in item 13. Separate lines are needed to set different components.

For boundary condition types 3 and 4, the X, Y, Z components conform to the X, Y, Z directions as specified in the COORD keyword for corner point geometry.


3. Name of the local grid refinement

4. A normal traction on this boundary, if the boundary condition type in item 2 is 1. X or Y or Z component of a given traction, if the boundary condition type in item 2 is 3. The component direction is set in item 13 below. Normal component or X component if type in item 2 is 3.

The sign of the traction is +ve for tension, -ve for compression. See the example below. (See also item 6 of the GEOMECH keyword.)

UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

DEFAULT: 0

5. Y component if type in item 2 is 3

UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

6. Z component if type in item 2 is 3

UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

7. A normal rock velocity on this boundary, if the boundary condition type in item 2 is 2

X or Y or Z component of a rock velocity vector, if the boundary condition type in item 2 is 4.

This is typically 0 to specify a rigid boundary.

Velocity vector orientation follows the orientation of the coordinate system.

UNITS: m/day (METRIC), ft/day (FIELD), cm/hr (LAB), m/day (PVT-M)

DEFAULT: 0

8. First block to be modified in the I direction

DEFAULT: 1 if an I- face

9. Last block to be modified in the I direction

DEFAULT: NX if an I+ face

10. First block to be modified in the J direction

DEFAULT: 1 if a J- face

11. Last block to be modified in the J direction

DEFAULT: NY if a J+ face

12. First block to be modified in the K direction
DEFaulT: 1 if a K- face

13. Last block to be modified in the K direction
DEFaulT: NZ if a K+ face

14. A character string that gives the component of traction or velocity if the boundary type in item 2 is either 3 or 4. This must be one of ‘X’, ‘Y’ or ‘Z’ and is required for type 3 of type 4 boundaries.
DEFaulT: None. It must be specified if item 2 is either 3 or 4

Examples

Example 1

```
-- set a 3000 psi total compressive traction on the x+ and y+ boundaries
-- note that the ‘K’ direction is positive downwards
STRESBCL
-- side type LGR total traction
‘I+’ 1 ‘LGR1’ -3000 /
‘J+’ 1 ‘LGR1’ -3000 /
‘K-’ 1 ‘LGR1’ -3000 /
```

Example 2

```
-- Velocities in X dir on I- and I+ boundaries are zero
-- Displacements in Y and Z dirs on I- and I+ boundaries are free
-- Velocities in Y dir on J- and J+ boundaries are zero
-- Displacements in X and Z dirs on J- and J+ boundaries are free
-- Velocities in Z dir on K- and K+ boundaries are zero
-- Displacements in X and Y dirs on K- and K+ boundaries are free
STRESBCL
-- side type LGR SXYZ SHEARS VXYZ range comp
‘I-’ 4 ‘LGR1’ 3* 0 6* ‘X’ /
‘I-’ 3 ‘LGR1’ 0 9* ‘Y’ /
‘I-’ 3 ‘LGR1’ 0 9* ‘Z’ /
‘I+’ 4 ‘LGR1’ 3* 0 6* ‘X’ /
‘I+’ 3 ‘LGR1’ 0 9* ‘Y’ /
‘I+’ 3 ‘LGR1’ 0 9* ‘Z’ /
‘J-’ 4 ‘LGR1’ 3* 0 6* ‘Y’ /
‘J-’ 3 ‘LGR1’ 0 9* ‘Z’ /
‘J-’ 3 ‘LGR1’ 0 9* ‘X’ /
‘J+’ 4 ‘LGR1’ 3* 0 6* ‘Y’ /
‘J+’ 3 ‘LGR1’ 0 9* ‘Z’ /
‘J+’ 3 ‘LGR1’ 0 9* ‘X’ /
‘K-’ 4 ‘LGR1’ 3* 0 6* ‘Z’ /
‘K-’ 3 ‘LGR1’ 0 9* ‘X’ /
‘K-’ 3 ‘LGR1’ 0 9* ‘Y’ /
‘K+’ 4 ‘LGR1’ 3* 1 6* ‘Z’ /
‘K+’ 3 ‘LGR1’ 0 9* ‘X’ /
‘K+’ 3 ‘LGR1’ 0 9* ‘Y’ /
```
Surface tension with respect to pressure

This provides the variation of surface tension with pressure. This should decrease with increasing pressure, falling to a low value near $P_{\text{sat}}$.

This is an alternative means of obtaining surface tensions for the $\text{MISCIBLE}$ option. If STVP is not entered the Macleod-Sugden correlation is used to obtain surface tensions.

A value of the reference surface tension, $s_0$, is still required. This may be entered with $\text{MISCSTR}$, $\text{MISCSTRP}$. If not, the largest value in any of the STVP tables is used.

STVP is a pressure table - one is required for each pressure table ($\text{PVTNUM}$) region.

A table consisting of two columns of data.

1. The pressure. Values should increase monotonically down the column.
   
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2. The corresponding surface tension.
   
   UNITS: dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT-M)

Example

```
STVP
  1 10
  100 8
  200 1
  300 0 /
```
**STWG**

**Water-gas surface tension versus pressure**

This keyword supplies tables of the water-gas interfacial tension versus pressure, which are used in the drift flux calculation for multisegment wells (see "The drift flux slip model" in the ECLIPSE Technical Description). If surface tension data is supplied for the table corresponding to the well’s PVT region number (see WELSPECS keyword item 11), the interfacial tension is taken from this table. If there is no surface tension data supplied for the well’s PVT table number, the interfacial tension is calculated from a built-in correlation.

The keyword should only be used if multisegment wells are present. It is not required by the SURFTENS option.

The data comprises NTPVT (see keyword TABDIMS in the RUNSPEC section) tables of water-gas surface tension versus pressure.

Each table consists of two records, each record terminated with a slash (/).

**Record 1**

A reference oil phase pressure.

This data item is not needed to calculate the interfacial tension in the drift flux model. The value entered here will not affect its results.

**UNITS:** barsa (METRIC), psia (FIELD), atma (LAB)

**Record 2**

A table consisting of two columns of data.

1. The pressure. Values should increase monotonically down the column.
   **UNITS:** barsa (METRIC), psia (FIELD), atma (LAB)

2. The corresponding water-gas interfacial tension.
   **UNITS:** N/m (METRIC), lbf/in (FIELD), dynes/cm (LAB)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see keyword TABDIMS in the RUNSPEC section).

**Example**

With NTPVT=1 and NPPVT ≥6:

```
STWG
  --  REFERENCE PRESSURE
  -- 
    5000 /
  -- PRESSURE  SURFACE TENSION
    1000  10.0
    2000  9.8
    3000  8.6
    4000  7.4
    5000  5.3
    6000  1.2 /
```
SUMTHIN  

Reduces the amount of SUMMARY data output

This keyword reduces the amount of data saved in the summary output files. This is achieved by only allowing one datum to be recorded for each SUMMARY keyword in a given time period. This time period is defined by the only item in this keyword. The first timestep in each period is output, with all subsequent output suppressed. Data are also recorded each report timestep regardless of the time since the last data were output.

**Note:** SUMTHIN will not change the timesteps used by ECLIPSE. This keyword does not change the way that the simulation is solved, it simply prevents the simulation from writing some data to the summary files. This means that data are not output at regular times. If you wish to achieve this use RPTONLY.

The SUMTHIN keyword contains one item, the time period for which only one set of data will be output. This should be followed by a slash (/). If the time period entered is zero, thinning will be switched off.

UNITs: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

See also the keyword RPTONLY.

**Example**

One set of data will be output in each 50 day period (in FIELD units). As a result the first timestep in each 50 day period will be written to the summary files, along with any report timesteps. This means that the first timestep between 0 and 50 is output, then the first timestep between 50 and 100 and so on.

```plaintext
SUMTHIN
50.0 /
```
**Initial surfactant concentrations**

This keyword sets the initial surfactant concentration in runs where the initial conditions are specified by enumeration.

The keyword should be followed by one real number for every grid block specifying the initial surfactant concentration. The data field should be terminated by a slash (/).

**UNITS:** kg/sm³ (METRIC), lb/stb (FIELD), gm/scc (LAB)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword **DIMENS**. Repeat counts may be used for repeated values (for example 115*0.18). Note that spaces must not be inserted on either side of the asterisk (*).

See also the keywords **PRESSURE, SGAS, RS, PBUB, RV, PDEW** and **SWAT** in the SOLUTION section. Refer to "Surfactant model" in the ECLIPSE Technical Description for more information.

**Notes**

- When adsorption is active, the concentration specified here will be honored and extra adsorbed surfactant is added to the rock so that it is in equilibrium with the input fluid concentration.

- If a non-zero concentration is specified in cells containing none of the host phase, the concentration will be automatically reset to zero. This behavior can be turned off using item 165 in OPTIONS.

**Example**

With NDIVIX=10, NDIVIY=1 and NDIVIZ=2 specified in RUNSPEC keyword **DIMENS**:

```
SURF
10*0.0
 9*0.0 0.5 /
```
**SURFACT** Enables the surfactant model

This indicates that the surfactant model is required (see "Surfactant model" in the *ECLIPSE Technical Description*).

The keyword has no associated data.

<table>
<thead>
<tr>
<th>Keywords</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>2179</td>
<td>2179</td>
</tr>
</tbody>
</table>
SURFACTW Enables modeling of changes of wettability in the surfactant model

This keyword enables the modeling of changes of wettability of the rock due to the accumulation of surfactant. It can be used in addition to the SURFACT keyword, or instead of it since it also activates the surfactant model.

Assuming that the usual SATNUM keyword defines oil-wet immiscible saturation functions (input with keywords SWFN, SOF2, SOF3 or SWOF), the user defines additional immiscible saturation functions with these same keywords and these are then taken to model the water-wet situation. The keyword SURFWNUM in the REGIONS section must be used to define the region (water-wet immiscible saturation function table) number of each grid block.

In calculating the immiscible relative permeabilities, a weighted average of $F$ times the oil-wet value and $(1- F)$ times the water-wet value is used. The fraction, $F$, is a function of the adsorbed surfactant concentration, and must be defined by tables input using the SURFADDW keyword in the REGIONS section; the number of the table used being that given in the SURFWNUM keyword.

See "Surfactant model" in the ECLIPSE Technical Description for further information.

The SURFACTW keyword has no associated data.
**SURFADDW  Tables of adsorbed surfactant concentration versus fractional saturation functions**

The data comprises \textit{NTSFUN} (item 1 of keyword \textit{TABDIMS}) tables of concentration of adsorbed surfactant versus the fraction of the oil-wet and water-wet saturation functions, to use in calculating the immiscible relative permeabilities when the wettability of the rock changes.

The tables describe the transition between oil-wet immiscible conditions and water-wet immiscible conditions as a function of the adsorbed surfactant concentration. The option to model wettability changes resulting from surfactant accumulation must be enabled with keyword \textit{SURFACTW} in the \textit{RUNSPEC} section.

Each table consists of 2 columns of data and should be terminated by a slash (/):

1. Concentration of adsorbed surfactant
   
   The values should increase monotonically down the column.  
   
   \textbf{UNITS}: kg/kg (\textit{METRIC}), lb/lb (\textit{FIELD}), gm/gm (\textit{LAB}), kg/kg (\textit{PVT-M})

2. Weighting of oil-wet to water-wet saturation function.
   
   A value of 1.0 implies that only the oil-wet saturation functions will be used (as defined by \textit{SATNUM}) and a value of 0.0 implies purely water-wet saturation functions (as defined by \textit{SURFWNUM}).

   The value should not be greater than 1.0 or less than 0.0

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than \textit{NSSFUN} (item 3 of keyword \textit{TABDIMS}).

\textbf{Note:} \textit{NTSFUN} tables must be supplied; however only the \textit{SURFWNUM} table numbers are used. If any table after the first is defaulted, then that table is copied from the previous table.

**Example**

With \textit{NTSFUN}=4 and \textit{NSSFUN} $\geq$ 4

```
SURFADDW
0  1.0
  2  1.0 / dummy table corresponding to a SATNUM table value
  / default the second table to the first table
  0  1.0
  20  1.0
  600  0.8
  1800  0.4/
  / default fourth table to the third table
```
Surfactant adsorption functions

The data comprises NTSFUN (see keyword TABDIMS) tables of surfactant adsorption functions, each terminated by a slash (/), describing the adsorption of surfactant by the rock formation. Each table consists of 2 columns of data:

1. The local surfactant concentration in the solution surrounding the rock.
   The values should increase monotonically down the column.
   
   **UNITS:** kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scc (LAB)

2. The corresponding saturated concentration of surfactant adsorbed by the rock formation.
   These values should increase monotonically down the column.

**Note:** The monotonicity constraint can be lifted by entering a non-zero value in item 76 of the OPTIONS keyword.

**UNITS:** kg/kg (METRIC), lb/lb (FIELD), gm/gm (LAB)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see keyword TABDIMS).

**CAUTION:** The first row in the adsorption tables should consist of a pair of zeroes.

See "Surfactant model" in the ECLIPSE Technical Description for further information.

Surfactant adsorption functions can also be specified using the ADSORB keyword.

**Example**

With NTSFUN=2 and NSSFUN ≥ 5

```
SURFADS .0000 .0000
 .0000 .0000
 .0003 .00005
 .0005 .00007
 .0008 .00008
 .010 .00008 /
 .0000 .00000
 .010 .00010 /
```

**Keywords**

ECLIPSE Reference Manual

2182
Surfactant capillary de-saturation functions

The data comprises NTSFUN (see keyword TABDIMS) tables of surfactant capillary de-saturation functions terminated by a slash (/). The de-saturation function describes the transition between immiscible conditions (low surfactant concentration) and miscibility (high surfactant concentration) as a function of the dimensionless capillary number. Each table consists of 2 columns of data:

1. The log of the capillary number ($\log_{10}(CAPN)$)
   - The values should increase monotonically down the column and should lie in the range -20 to 20.

2. The miscibility function at the value of the log capillary number.
   - A value of 0 implies immiscible conditions and a value of 1 miscible conditions.
   - The values should not be greater than 1 or less than 0

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see keyword TABDIMS).

**Note:** NTSFUN tables must be supplied; however, only the SATNUM table numbers are used, and any tables referred to by SURFNUM and not by SATNUM is ignored. If any table after the first is defaulted, then that table is copied from the previous table.

See "Surfactant model" in the *ECLIPSE Technical Description* for further information.

**Example**

With NTSFUN=2 and NSSFUN ≥4

```
SURFCAPD
-10  0.0
-3   0.0
-2   1.0
10   1.0   /
/ default the second table to the first table
```
**SURFESAL**  
**Effective salinity coefficient for surfactant**

This keyword may only be used when the multi-component brine model (keyword `BRINE` in conjunction with keyword `ECLMC` in the `RUNSPEC` section) is active in conjunction with the surfactant option (keyword `SURFACT` in the `RUNSPEC` section).

The `SURFESAL` keyword activates the effective salinity calculation for surfactant properties and defines the parameter, $\alpha^s$, in the expression for the effective salinity

$$ C_{SE} = \frac{C_-}{1 - \alpha^s X^s} $$

Eq. 3.200

where

- $C_{SE}$ denotes the effective salinity for the surfactant
- $C_-$ denotes the anion concentration
- $X^s$ denotes the fraction of anionic surfactant sites occupied by divalent cations
- $\alpha^s$ denotes the effective salinity parameter

See "Brine tracking" in the *ECLIPSE Technical Description* for details of this option.

The keyword should be followed `NTSFUN` records of data (see item 1 in keyword `TABDIMS`), each containing a single non-negative real number. Each record must be terminated by a slash (/).

**Example**

For `NTSFUN=2`:

```plaintext
SURFESAL
  -- alpha
  0.80 /
  0.65 /
```
Surfactant miscible region numbers

This keyword specifies the saturation table numbers used by the surfactant model to calculate the relative permeabilities when a high surfactant concentration implies the oil and water are miscible. The usual \texttt{SATNUM} keyword is used to specify the immiscible saturation functions.

The keyword should be followed by one integer for every grid block in the current input box, specifying the miscible (high surfactant) saturation function region to which it belongs. The region number should not be less than 1 or greater than \texttt{NTSFUN} (see \texttt{RUNSPEC} section keyword \texttt{TABDIMS}). The data must be terminated by a slash (/).

The saturation functions are specified using the \texttt{SWFN}, \texttt{SOF2}, \texttt{SOF3} or \texttt{SWOF} keywords. A non-zero capillary pressure entered in the \texttt{SWFN} or \texttt{SWOF} keyword is ignored.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

The \texttt{SURFNUM} keyword should be specified when the \texttt{SURFACT} option has been specified. If this keyword is undefined or only partially specified when the \texttt{SURFACT} option has been specified, the unspecified miscible saturation region numbers will default to 1.

\textbf{Note:} When end-point scaling has been specified via the \texttt{ENDSCALE} option, no end-point scaling will be applied to the saturation table used for the miscible saturation functions.

\textbf{Example}

With \texttt{NTSFUN}=4 specified in keyword \texttt{TABDIMS}; \texttt{NDIVIX}=10, \texttt{NDIVIY}=10 and \texttt{NDIVIZ}=1 specified in \texttt{RUNSPEC} keyword \texttt{DIMENS}; and no input \texttt{BOX} set

```
SATNUM
  50*1 50*2 /
SURFNUM
  50*3 50*4 /
```
**SURFOPTS**  

**Surfactant options**

This keyword sets optional parameters for the ECLIPSE 300 surfactant model. See the "Surfactant model" in the *ECLIPSE Technical Description* for further information.

1. The minimum water saturation for surfactant effects.

   DEFAULT: \(1 \times 10^{-6}\)

   For cells with small water saturations, a small amount of surfactant will give a high concentration of surfactant in the water phase, which could have a large effect on the capillary pressure. To prevent numerical errors effecting the solution, surfactant concentrations are disregarded for cells where the minimum water saturation is below a threshold given by this item.

**Example**

```
SURFOPTS
2.0e-6 /
```
SURFROCK  Specifies rock properties for the surfactant model

The data comprises NTSFUN (see keyword TABDIMS) tables specifying the rock properties which are required for the Surfactant model. Each table should be terminated by a slash (/) and requires the following items of data:

1. The adsorption index to be used for this rock type.
   Possible values for the adsorption index are 1 or 2.
   If a value of 1 is selected, then the surfactant adsorption isotherm is retraced whenever the local surfactant concentration in the solution decreases.
   If a value of 2 is selected, then no surfactant desorption may occur.

2. The mass density of this rock type at reservoir conditions.
   This quantity is used in the calculation of the surfactant loss due to adsorption.
   UNITS: kg/m³ (METRIC), lb/rb (FIELD), gm/cc (LAB)

Notes

- If the Polymer Flood Model is being used, then the mass density is ignored. The rock mass density is taken from the PLYROCK keyword.
- When non-zero initial surfactant concentrations are specified via the SURF or TVDP keywords, the concentration specified in that keyword will be honored and extra adsorbed surfactant is added to the rock so that it is in equilibrium with the input fluid concentration.

See "Surfactant model" in the ECLIPSE Technical Description for further information.

Example

With NTSFUN=2

```
SURFROCK
2  2340  /
1  2520  /
```
**SURFST**

**Water/oil surface tension versus surfactant concentration**

This keyword supplies tables of water-oil surface tension as a function of surfactant concentration in the water. The keyword is required for surfactant runs and is used to control the surfactant relative permeability and capillary pressure model. See “Surfactant model” in the *ECLIPSE Technical Description* for further information.

The data comprises NTPVT (see keyword **TABDIMS**) tables of water-oil surface tension versus surfactant concentration. Each table consists of two columns of data, terminated with a slash (/).

1. The surfactant concentration.
   - Values should increase monotonically down the column.
   - UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scv (LAB), kg/sm$^3$ (PVT-M)

2. The corresponding water-oil surface tension.
   - UNITS: N/m (METRIC), lbf/in (FIELD), dynes/cm (LAB), N/m (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see keyword **TABDIMS**).

**Example**

With NTPVT=1 and NPPVT ≥5

```
SURFST
--  Concentration   Surface Tension
   --
   0.00   10.0
   0.01   5.0
   0.015  2.0
   0.02   1.0
   0.025  0.1 /
```

**Keywords**
SURFSTES Water/oil surface tension versus surfactant and salt concentrations

This keyword should only be used for surfactant studies when the salt sensitivity option has been activated (keyword BRINE in the RUNSPEC section). See the "Surfactant model" in the ECLIPSE Technical Description for further information.

The data comprises NTPVT (see item 2 in keyword TABDIMS) items of oil-water surface tension, each terminated by a slash (/). These describe the effect on the oil-water surface tension of increasing the concentration of surfactant and salt in the solution. Each table consists of up to NPPVT (see RUNSPEC keyword TABDIMS) sets of data records. Each set of records within a table consists of up to NPPVT records of data items to allow the specification of a full two dimensional solution surface tension function with respect to surfactant concentration and effective salinity. The first record of each elementary set of records contains the following items of data:

1. The surfactant concentration.
   Values should increase monotonically down the column.
   UNITS: kg/sm³ (METRIC), lb/stb (FIELD), gm/sec (LAB), kg/sm³ (PVT-M)

2. The corresponding value of the oil-water surface tension at the surfactant concentration given by item 1 and a salt concentration given by item 1 in the corresponding ESSNODE table.
   UNITS: N/m (METRIC), lbf/in (FIELD), dynes/cm (LAB), N/m (PVT-M)

The data for each table is terminated by a null record (that is a record with no data before the terminating slash).

See the "Surfactant model" in the ECLIPSE Technical Description for further information.

Example
With NTPVT=2 and NPPVT ≥3

```
SURFSTES
---
<table>
<thead>
<tr>
<th></th>
<th>Surface Tension</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>2.00E-2</td>
</tr>
<tr>
<td></td>
<td>1.99E-2</td>
</tr>
<tr>
<td></td>
<td>1.98E-2</td>
</tr>
<tr>
<td>5</td>
<td>2.60E-5</td>
</tr>
<tr>
<td></td>
<td>2.60E-5</td>
</tr>
<tr>
<td></td>
<td>2.60E-5</td>
</tr>
<tr>
<td>10</td>
<td>8.00E-6</td>
</tr>
<tr>
<td></td>
<td>7.00E-6</td>
</tr>
<tr>
<td></td>
<td>6.00E-6</td>
</tr>
<tr>
<td>/</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>2.00E-2</td>
</tr>
<tr>
<td></td>
<td>1.99E-2</td>
</tr>
<tr>
<td></td>
<td>1.98E-2</td>
</tr>
<tr>
<td>6</td>
<td>2.60E-5</td>
</tr>
<tr>
<td></td>
<td>2.60E-5</td>
</tr>
<tr>
<td></td>
<td>2.60E-5</td>
</tr>
<tr>
<td>12</td>
<td>8.00E-6</td>
</tr>
<tr>
<td></td>
<td>7.00E-6</td>
</tr>
<tr>
<td></td>
<td>6.00E-6</td>
</tr>
<tr>
<td>/</td>
<td></td>
</tr>
</tbody>
</table>
```
**Surfactant solution viscosity function**

The data comprises NTPVT (see keyword `TABDIMS`) tables of surfactant viscosity functions, each terminated by a slash (/), describing the effect on the viscosity of pure water of increasing the concentration of surfactant in solution. Each table consists of 2 columns of data:

1. The surfactant concentration in the solution.
   - The values should increase monotonically down the column.
   - UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/sec (LAB)

2. The solution water viscosity at this surfactant concentration and the reference pressure (entered with the `PVTW` keyword).
   - UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)
   - If the Polymer Flood Model is active, then the viscosity input in the `SURFVISC` keyword will be multiplied by the polymer viscosity multiplier supplied in the `PLYVISC` or `PLYVISCs` keywords.

**Note:** The viscosity entered in this keyword at zero surfactant concentration should be equal to the viscosity entered in keyword `PVTW` (or `PVTWSALT`, whichever is used).

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see keyword `TABDIMS`).

See "Surfactant model" in the *ECLIPSE Technical Description* for further information.

**Example**

With NTPVT=1 and NPPVT ≥3

```
SURFVISC
  .00  0.34
  .01  0.38
  .02  0.40 /
```

**Keywords**

2190
SURFWNUM  Water-wet saturation function region and oil/water fraction table numbers

The keyword should be followed by one integer for every grid block specifying the (high-salinity) water-wet saturation function region to which it belongs. The region number should not be less than 1 or greater than NTSFUN (see item 1 of keyword TABDIMS). The data must be terminated by a slash (/).

The (high salinity) water-wet saturation function region number specifies the saturation table (input using SWFN, SOF3 and related keywords in the PROPS section) to be used to calculate relative permeabilities and capillary pressures in each grid block.

If the LOWSALT option has also been specified, the water-wet curves are computed as a weighted average between the low salinity water-wet saturation functions (LSLTWNUM region numbers) and the high salinity water-wet saturation functions (SURFWNUM region numbers). In this case, the salinity weighting factors are taken from the LSALTFNC keyword according to the region numbers specified by the LWSLTNUM keyword.

The table number specified here also defines the associated table number for the SURFADDW keyword. This specifies the weighting between immiscible water-wet and oil-wet saturation functions as a function of adsorbed surfactant concentration. The SURFWNUM keyword can be used only if the SURFACTW keyword has been entered in the RUNSPEC section.

Grid blocks are ordered with the X-axis index cycling fastest, followed by the Y- and Z-axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

**Note:** The SURFWNUM keyword or its alias HWSNUM must be specified if the SURFACTW option has been specified.

The LSLTWNUM keyword or its alias LWSNUM must be specified if the LOWSALT and SURFACTW options have been specified.

The LWSLTNUM keyword or its alias LSNUM must be specified if the LOWSALT option has been specified.

**Example**

With NTGFUN=4 specified in keyword TABDIMS; NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS; and no input BOX set:

```
SURFWNUM
144*3 96*4 /
```
Initial water saturations

The keyword should be followed by one real number for every grid block specifying the initial water saturation. The data field should be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest (from 1 to NDIVIX), followed by the Y-axis index (from 1 to NDIVIY), and finally the Z-axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*).

This keyword may be used to specify the initial solution explicitly, as an alternative to equilibration using EQUIL. It will be expected if the PRESSURE keyword is present, indicating explicit initialization, and if a water phase is present.

For the Mixed Hydrostatic Equilibration option, initial saturations should be specified together with items 1 and 2 of the EQUIL keyword and the NEI keyword. For more information, refer to "Initializing the study" in the ECLIPSE Technical Description.

When using explicit initialization, you must ensure that the initial solution is stable and physically reasonable.

Example

With NDIVIX=8, NDIVIY=6, NDIVIZ=5 specified in RUNSPEC keyword DIMENS:

```
SWAT
48*0
48*0.04
48*0.2
48*0.7
48*1       /
```
SWATINIT

Initial water saturations for capillary pressure scaling

This keyword allows you to input a water distribution and to scale the water-oil capillary pressure curves such that this water distribution is honored in the equilibrated initial solution. Note that any water saturation value that cannot be honored because it is below the water contact is reset to the maximum water saturation in the subsequent equilibration calculation.

The scaled maximum water capillary pressures ($PPCW$) can be output to the print file. This can be used to check for unphysically high maximum capillary pressures, which can be limited by using the $PPCWMAX$ keyword.

In ECLIPSE 100, to output $PPCW$ data, use the mnemonic $SWATINIT$ in the $RPTSOL$ keyword. In ECLIPSE 300, to output $PPCW$ data, use the mnemonic $PPCW$ in the $RPTSOL$ keyword.

The $SWATINIT$ keyword may only be used if the end point scaling option is active (keyword $ENDSCALE$ in the $RUNSPEC$ section).

Note: If the $ENDSCALE$ keyword is not present in the $RUNSPEC$ section, ECLIPSE 100 will assume that it is, and use the $ENDSCALE$ defaults.

The keyword should be followed by one real number for every grid block in the current input box, specifying the required water saturation. The data field should be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest (from 1 to $NDIVIX$), followed by the Y-axis index (from 1 to $NDIVIY$), and finally the Z-axis index (from 1 to $NDIVIZ$). $NDIVIX$, $NDIVIY$ and $NDIVIZ$ are specified in $RUNSPEC$ keyword $DIMENS$. Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*).

See also the $PCW$ and $PPCWMAX$ keywords in the $PROPS$ section and the $EQUIL$ keyword in the $SOLUTION$ section.

Further information can be found in "Matching initial water distribution" in the ECLIPSE Technical Description.

Notes

If a grid block is given a water saturation less than 1.0 below the oil water contact (where $P_c=0$), then the $SWATINIT$ saturation is not honored. The $SWATINIT$ saturation below the oil water contact will be honored under item 74 in $OPTIONS$ and item 143 in $OPTIONS3$ for a different scaling. The capillary pressure is given by:

\[
P_c = (PCW - P_{cmin}) \left( \frac{P_{cl} - P_{cmin}}{P_{cmax} - P_{cmin}} \right) + P_{cmin}
\]

Eq. 3.201

for switch equal 5.

and

\[
P_c = (P_{cmax} - PCW) \left( \frac{P_{cl} - P_{cmin}}{P_{cmax} - P_{cmin}} \right) + PCW
\]

Eq. 3.202

for switch equal 7.
where

\( P_{ct} \)  Capillary pressure from the table

\( P_{cmax} \)  Maximum \( P_c \) in the table

\( P_{cmin} \)  Minimum \( P_c \) in the table

PCW  Maximum / Minimum \( P_c \) from the PCW data.

- If a cell is given saturation corresponding to a zero capillary pressure (typically 1.0) above the contact, then the \( P_c \) curve cannot be scaled to honor the saturation, hence the \( P_c \) curve is left unscaled.

- If the SWATINIT saturation is less than the connate water saturation for a cell, the initial water saturation will be reset to the connate saturation and the capillary pressure will be scaled according to this value.

- The SWATINIT saturation may not be exactly honored if the fine grid block equilibration option is used in the EQUIL keyword.

- If the PCW keyword has been used, then this is ignored in regions where SWATINIT is set.

- If the Leverett J-function calculation for the water-oil capillary pressure has been activated through the JFUNC keyword, then J-function calculation is ignored in regions where SWATINIT is set.

- The SWATINIT saturation is not honored if the capillary pressure curve is a constant. The input capillary pressure curve needs to be monotonically decreasing with increasing water saturation.

**Example**

With \( \text{NDIVIX}=20, \text{NDIVIY}=6 \) and \( \text{NDIVIZ}=5 \) specified in RUNSPEC keyword DIMENS:

```
SWATINIT
  120*0
  120*0.1
  120*0.5
  120*0.7
  120*1 /
```
The SWCR and ISWCR keywords specify the critical water saturation (that is the largest water saturation for which the water relative permeability is zero) within each grid cell. The water flow across each grid face is calculated from a transformed water relative permeability curve obtained by linearly scaling the tabulated relative permeability data between the new critical water saturation defined using an SWCR keyword, and the new maximum water saturation (see under keyword SWU in the PROPS section). The SWCR keyword also permits scaling of the relative permeability table used in computing the flow of water between grid cells and well connections.

Seven keywords are provided to allow the critical water saturation for each grid cell to be scaled independently. These keywords are:

- SWCR: Critical water saturation for the non-directional saturation function tables
- SWCRX: Critical water saturation for the +X face of each grid cell
- SWCRX-: Critical water saturation for the -X face of each grid cell
- SWCRY: Critical water saturation for the +Y face of each grid cell
- SWCRY-: Critical water saturation for the -Y face of each grid cell
- SWCRZ: Critical water saturation for the +Z face of each grid cell
- SWCRZ-: Critical water saturation for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the critical water saturations for flows in the +R, -R, +T, -T, +Z, -Z directions respectively.

None of the SWCR keywords should be used if water is not present in the model. The keyword ENDSCELE should be specified in the RUNSPEC section.

If both directional and irreversible end-point scaling is requested, using switches 'DIRECT' and 'IRREVERS' in the ENDSCELE keyword in RUNSPEC, the critical water saturations for water flow across the +X and -X faces of a grid cell are set using the keywords SWCRX, SWCRX-. Similarly SWCRY, SWCRY- are used for the +Y, -Y faces and the keywords SWCRZ, SWCRZ- for the +Z, -Z faces. Scaling of the well connection relative permeability tables for any well connections in the grid cell are performed using the SWCR keyword.

When the end-point scaling is requested to be directional and reversible, using the switch 'DIRECT' but not 'IRREVERS ' in ENDSCELE, the keyword SWCRX defines the critical water saturations for the +X and -X faces of each grid cell. Keywords SWCRY, SWCRZ are used for both the +Y, -Y faces and the +Z, -Z faces of each grid cell respectively. Scaling of well connection relative permeability tables uses data entered under the SWCR keyword.

If ENDSCELE is specified in RUNSPEC without the switch 'DIRECT', then none of the directional forms of the SWCR keyword are allowed. The SWCR keyword then defines the new critical water saturation for flow out of each face of the grid cell and for water flows between the grid cell and well connections.

Each SWCR keyword should be followed by one real number for each grid block in the current input box specifying the critical water saturation within the grid block. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash(/).
Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If any of the keywords SWCR, SWCRX, SWCRX-, SWCRY, SWCRZ, SWCRZ- are omitted in a model where they could be accepted then the critical water saturation for a water relative permeability calculation defaults to the value used in the appropriate relative permeability table.

If the Hysteresis model (switch 'HYSTER' in keyword SATOPTS) is active then the SWCR data scales the drainage curve. The imbibition curve can be scaled by using the equivalent ISWCR (ISWCRX, ISWCRX-, ...) keyword.

If the ODD3P relative permeability and capillary pressure hysteresis model has been selected, the primary end-point scaling is specified using the PSORG, PSGRO, PSORW, PSWRO, PSGRW and PSWRG keywords and the hysteresis end-point scaling is specified using the HSORG, HSGRO, HSORW, HSWRO, HSGRW and HSWRG keywords.

See also keywords SWU, SWL, ENPTVD in the PROPS section.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```plaintext
-----RUNSPEC section
DIMENS
1 7 3 /
ENDSCALE
/
-----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
 1 1 2 7 1 3 /
SWCR
5*0.22 10*0.17 3*0.19 /
ENDBOX
```

ECLIPSE 300

Three phase water saturation functions

This keyword should only be used when the IKU three-phase relative permeability model has been requested using the IKU3P keyword or when the ODD3P three-phase relative permeability and capillary pressure hysteresis model has been requested using the ODD3P keyword or when the water BAKER three-phase relative permeability and capillary pressure hysteresis model has been requested using the BAKER1 or BAKER2 keywords in the PROPS section.

The data consists of tables of two-phase water saturation functions, each terminated with a slash (/). Each table consists of 4 columns of data:

Column:

1. The water saturation.
   Values should be between 0 and 1 and should increase monotonically down the column.

2. The corresponding water in oil relative permeability measured in the absence of gas.
   Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0.

3. The corresponding water in gas relative permeability measured in the absence of oil.
   Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0.
   For the ODD3P option, the relative permeability values between the critical and maximum saturations are not used and may be defaulted.

4. The corresponding water-oil capillary pressure.
   Values should be level or decreasing down the column.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

The number of tables supplied should be equal to the number of saturation tables in the run, specified in item 1 of keyword TABDIMS in the RUNSPEC section.

The number of rows should be less than or equal to the maximum number of rows of saturation data specified in item 3 of TABDIMS.

Note: The critical saturation for each relative permeability function is established by determining the highest saturation in the table for which the relative permeability is notionally zero, that is, less than a small threshold value. Relative permeability values which are lower than this threshold value will be treated as zero. For ECLIPSE 100 and ECLIPSE 300 the default threshold values are $1.0 \times 10^{-6}$ and $1.0 \times 10^{-20}$ respectively. These values may be modified using the TOLCRIT keyword.

See also "Saturation functions" in the ECLIPSE Technical Description.

Example

With one saturation table:

```
SWF3
.22 .00 .00 7.0
.30 .00 .07 4.0
.40 .05 .15 3.0
```
| .50 | .12 | .24 | 2.5 |
| .60 | .21 | .33 | 2.0 |
| .80 | .44 | .65 | 1.0 |
| .90 | .76 | .83 | .5 |
| 1.00 | 1.00 | 1.00 | .0 |

Keywords

2198
SWF32D  

**2D table of water relative permeability**

This keyword is an alternative to the SWFN keyword, allowing the three-phase relative permeability of water to be input directly as a function of oil and gas saturations in a two dimensional table.

The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) tables of water relative permeability, tabulated against both oil and gas saturations, each terminated by a slash (/).

Each table consists of a number of records:

**Record 1:**  
The oil saturation values

Values should be between 0 and 1 and should increase monotonically. The number of oil saturation values should not exceed NSSFUN (see item 3 of keyword TABDIMS).

**Subsequent records (a maximum of NSSFUN records):**  
The gas saturation.

Followed by the water relative permeability values for this gas saturation at all the corresponding oil saturations entered in Record 1.

The water relative permeabilities should be monotonically decreasing, with a final value of zero. The number of water relative permeability values should not exceed the number of oil saturation values entered in Record 1. The record may be truncated at any point provided that the last relative permeability value is zero.

The columns of data must also comply with the following constraints:

- The gas saturation (the first item in the second and subsequent records) should increase monotonically down the column. The water relative permeability for each oil saturation should decrease monotonically down the column and should end in a zero value.

- The relative permeability must also be zero for any point on the table in which \( S_{oil} + S_{gas} \geq 1 \).

- Default values may not be placed within the table.

The entire table may be defaulted provided that the table is not the first. Defaulted tables are replaced with a copy of the previous table.

**ECLIPSE 100**  
The water/oil capillary pressure must be provided using the PCW32D keyword.

**ECLIPSE 300**  
The water/oil capillary pressure can be provided using the SWFN keyword. The relative permeability data entered with the SWFN keyword will be ignored.

The keyword cannot be used with the end-point scaling, hysteresis and miscible options.

**Example**

With NTSFUN=1 and NSSFUN ≧11

```
| SWF32D |                          | SOIL | 0.22 0.27 0.32 0.37 0.42 0.47 0.52 0.57 0.72 0.77 0.78 / |
|--------|---------------------------|------|---------------------|---------------------|
|        | --SGAS                    |      | 0.00 1.000 0.625 0.345 0.207 0.113 0.083 0.035 0.023 0.002 0.001 0.000 / |
|        | 0.05 0.555 0.337 0.210 0.110 0.078 0.047 0.021 0.004 0.001 0.000 / |
|        | 0.10 0.390 0.212 0.106 0.074 0.042 0.019 0.003 0.002 0.000 / |
|        | 0.15 0.215 0.103 0.069 0.036 0.017 0.003 0.002 0.001 0.000 / |
|        | 0.20 0.100 0.065 0.031 0.015 0.002 0.002 0.001 0.000 / |
|        | 0.25 0.060 0.025 0.014 0.002 0.001 0.001 0.000 / |
|        | 0.30 0.020 0.012 0.001 0.001 0.001 0.000 / |
```
<table>
<thead>
<tr>
<th>0.35</th>
<th>0.010</th>
<th>0.001</th>
<th>0.001</th>
<th>0.001</th>
<th>0.000</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.40</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>
The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) tables of water saturation functions, each terminated by a slash (/).

Each table consists of 3 columns of data:

1. The water saturation.
   Values should be between 0 and 1 and should increase monotonically down the column.

2. The corresponding water relative permeability.
   Values should be between 0 and 1 and should be level or increasing down the column.
   The first value in the column must be 0.

3. The corresponding water-oil capillary pressure.
   Values should be level or decreasing down the column. Note, however, that if SWATINIT is used, then the water-oil capillary pressures must be monotonically decreasing.
   In gas-water systems (that is with no active oil phase), the water-gas capillary pressure should be entered here.

   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

Default values (represented by 1*) may be inserted as required in columns 2 and 3. When the table is read in, defaults are replaced by values computed by linear interpolation.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see item 3 of keyword TABDIMS).

The entire table may be defaulted provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

The keyword families

(i) SWOF, SGOF, SLGOF

(ii) SWFN, SGFN, SGWFN, SOF2, SOF3, SOF32D

provide two alternative formats for inputting the saturation functions. Do not mix keywords from the two families in the same run. In gas-water runs, if you are using the ECLIPSE 100 Miscible Flood or Solvent options, the saturation functions must be input with keywords from family (ii).

In gas-water runs the gas and water saturation functions may be input together using the SGWFN keyword as an alternative to using both SGFN and SWFN.

Note: The critical saturation for each relative permeability function is established by determining the highest saturation in the table for which the relative permeability is notionally zero, that is, less than a small threshold value. Relative permeability values which are lower than this threshold value will be treated as zero. For ECLIPSE 100 and ECLIPSE 300 the default threshold values are $1.0 \times 10^{-6}$ and $1.0 \times 10^{-20}$ respectively. These values may be modified using the TOLCRIT keyword.
Example

With $\text{NTSFUN}=2$ and $\text{NSSFUN} \geq 8$:

<table>
<thead>
<tr>
<th>SWFN</th>
<th>.2200</th>
<th>.0000</th>
<th>7.0000</th>
</tr>
</thead>
<tbody>
<tr>
<td>.3000</td>
<td>.0700</td>
<td>4.0000</td>
<td></td>
</tr>
<tr>
<td>.4000</td>
<td>.1500</td>
<td>3.0000</td>
<td></td>
</tr>
<tr>
<td>.5000</td>
<td>.2400</td>
<td>1*</td>
<td></td>
</tr>
<tr>
<td>.6000</td>
<td>.3300</td>
<td>2.0000</td>
<td></td>
</tr>
<tr>
<td>.8000</td>
<td>.6500</td>
<td>1.0000</td>
<td></td>
</tr>
<tr>
<td>.9000</td>
<td>.8300</td>
<td>1*</td>
<td></td>
</tr>
<tr>
<td>1.0000</td>
<td>1.0000</td>
<td>.0000</td>
<td>/</td>
</tr>
<tr>
<td>.18</td>
<td>.00</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>.32</td>
<td>.07</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>.50</td>
<td>.31</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>.60</td>
<td>.38</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>.80</td>
<td>.57</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>1.00</td>
<td>1.00</td>
<td>0</td>
<td>/</td>
</tr>
</tbody>
</table>

Note: Slow run times may result if the relative permeability changes abruptly over a small saturation interval, especially if IMPES is used. ECLIPSE honors the data input and does not attempt to overcome convergence problems by retabulating over equally spaced saturation intervals. If chopped relative permeability data is used to minimize dispersion, ECLIPSE performs best in fully-implicit mode if the relative permeability rises steeply from zero (immobility) rather than from a small value.
**Scaled critical water saturations in gas**

This keyword specifies the critical water saturation in gas within each grid cell. This is available when the IKU3P, BAKER1 or BAKER2 three phase relative permeability model is being used and the two-phase water-gas relative permeabilities are specified using the SWF3 keyword. The critical water saturation in oil is obtained from SWCR.

Seven keywords are provided to allow the critical water saturation in gas for each grid cell to be scaled independently. These keywords are:

- **SWGCR**: Critical water saturation in gas for the non-directional saturation function tables
- **SWGCRX**: Critical water saturation in gas for the +X face of each grid cell
- **SWGCRX-**: Critical water saturation in gas for the -X face of each grid cell
- **SWGCRY**: Critical water saturation in gas for the +Y face of each grid cell
- **SWGCRY-**: Critical water saturation in gas for the -Y face of each grid cell
- **SWGCRZ**: Critical water saturation in gas for the +Z face of each grid cell
- **SWGCRZ-**: Critical water saturation in gas for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the critical water saturations in gas for flows in the +R, -R, +T, -T, +Z, -Z directions respectively.

None of the SWGCR keywords should be used if water is not present in the model. The keyword ENDScale should be specified in the RUNSPEC section.

If both directional and irreversible end-point scaling are requested, using switches 'DIRECT' and 'IREVERS' in the ENDScale keyword in RUNSPEC, the critical water saturations in gas for flow across the +X and -X faces of a grid cell are set using the keywords SWGCRX and SWGCRX-. Similarly the keywords SWGCRY and SWGCRY- are used for the +Y and -Y faces and the keywords SWGCRZ and SWGCRZ- for the +Z and -Z faces respectively. Scaling of the well connection relative permeability tables for any well connections in the grid cell is performed using the values specified via the SWGCR keyword.

When the end-point scaling is requested to be directional and reversible, using the switch 'DIRECT' but not 'IREVERS' in ENDScale, the keyword SWGCRX defines the critical water saturations in gas for the +X and -X faces of each grid cell. Keywords SWGCRY and SWGCRZ are used for both the +Y and -Y faces and the +Z and -Z faces of each grid cell respectively. Scaling of the well connection relative permeability tables for any well connections in the grid cell is performed using the values specified via SWGCR keyword.

If ENDScale is specified in RUNSPEC without the switch 'DIRECT', none of the directional forms of the SWGCR keyword are allowed. The SWGCR keyword then defines the critical water saturation in gas for flow out of each face of the grid cell and for flows between the grid cell and well connections.

Each SWGCR keyword should be followed by one real number for each grid block in the current input box specifying the critical water saturation in gas within the grid block. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash(/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.
If any of the keywords SWGCR, SWGCRX, SWGCRX-, SWGCRY, SWGCRY-, SWGCRZ or SWGCRZ- are omitted in a model where they could be accepted, the critical water saturation in gas for a water relative permeability calculation defaults to the value specified in the appropriate relative permeability table.

If the Hysteresis model (switch 'HYSTER' in keyword SATOPTS) is active, the SWGCR data scales the drainage curve. The imbibition curve can be scaled by using the corresponding ISWGCR (ISWGCRX, ISWGCRX-, ...) keywords. (The IKU3P three phase relative permeability model does not support hysteresis.)

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```plaintext
-----RUNSPEC section
DIMENS
1 7 3 /
ENDSCALE

-----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3 /
SWGCR
  5*0.1 10*0.2 3*0.1 /
ENDBOX
```
The SWINGFAC keyword turns on the gas field operations model and defines the swing factors and profile factors for the FIELD. (If you want to apply separate gas supply contracts to two or more groups, use keyword GSWINGF instead.) The following features of the model are activated when the keyword is first entered:

- Group and field gas rate targets are applied to sales gas rates instead of gross gas production rates.
- The delivery capacity is calculated at the beginning of each timestep.
- In ECLIPSE 100, any automatic compressors (specified with keyword GASFCOMP or NETCOMPA) will be turned on and off as required to meet the field gas target.

The swing factors and profile factors themselves are used to set the field gas production target and to adjust the DCQ when the simulation is advanced using the keyword GASYEAR or GASPERIO instead of DATES, TIME or TSTEP. The FIELD is automatically placed under Guide Rate Control to meet any gas rate targets imposed, unless Prioritization is requested with the keyword GCONPRI.

See "Gas field operations model" in the ECLIPSE Technical Description for further details.

The SWINGFAC keyword is followed by 24 numbers, terminating with a slash (/).

- The first 12 numbers are the swing factors for each month in sequence from January to December.
- The second 12 numbers are the profile factors for each month in sequence from January to December.

A warning is printed if the profile factor for any month is greater than that month’s swing factor. If this is so, obeying the swing requirement may not guarantee the field could produce its target rate profile.

### Normalized profile factors

In order to produce the required ACQ for a given DCQ, the profile factors should be normalized such that the sum over all months of \{profile factor \times number of days in month\} = 365.25, with February allocated 28.25 days.

If this “normalization requirement” is obeyed, then when leap years are taken into account every fourth year, the average annual production at a constant DCQ is DCQ \times 365.25, which is the ACQ. A warning is printed if the profile factors are not normalized.

### Example

```
SWINGFAC
-- JAN  FEB  MAR  APR  MAY  JUN  JUL  AUG  SEP  OCT  NOV  DEC
12*1.67  
2*1.5036 3*1.0024  3*0.5012  3*1.0024  1.5036 /
```
Scaled connate water saturations

The different forms of this keyword specify the connate water saturation (that is the smallest water saturation in a water saturation function table) within each grid cell. The oil-water (or gas-water) capillary pressure used in computing the water flow across each grid face is calculated from a transformed water capillary pressure curve obtained by linearly scaling the tabulated capillary pressure data between the new connate water saturation defined using an SWL keyword, and the new maximum water saturation (see under the keyword SWU in the PROPS section). The SWL keyword also permits scaling of the water capillary pressure table used in computing the initial water saturation distribution.

Seven keywords are provided to allow the connate water saturation for each grid cell to be scaled independently. These keywords are:

- **SWL**: Connate water saturation for the non-directional saturation function tables
- **SWLX**: Connate water saturation for the +X face of each grid cell
- **SWLX-**: Connate water saturation for the -X face of each grid cell
- **SWLY**: Connate water saturation for the +Y face of each grid cell
- **SWLY-**: Connate water saturation for the -Y face of each grid cell
- **SWLZ**: Connate water saturation for the +Z face of each grid cell
- **SWLZ-**: Connate water saturation for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the connate water saturations for flows in the +R, -R, +T, -T, +Z, -Z directions respectively.

None of the SWL keywords should be used if water is not present in the model. The keyword ENDSCALE should be specified in the RUNSPEC section.

If both directional and irreversible end-point scaling is requested, using switches 'DIRECT' and 'IRREVERS' in ENDSCALE, connate water saturations for water flow across the +X and -X faces of a grid cell are set using the keywords SWLX, SWLX-. Similarly SWLY, SWLY- are used for the +Y, -Y faces and the keywords SWLZ, SWLZ- for the +Z, -Z faces. Scaling of the oil-water (or gas-water) capillary pressure tables in the equilibration algorithm are performed using the SWL keyword.

When the end point scaling is requested to be directional and reversible, using the switch 'DIRECT' but not 'IRREVERS' in ENDSCALE, the keyword SWLX defines the connate water saturations for the +X and -X faces of each grid cell. Keywords SWLY, SWLZ are used for both the +Y, -Y faces and the +Z, -Z faces of each grid cell respectively. Scaling of oil-water (or gas-water) capillary pressure data for the equilibration algorithm is done with the SWL keyword.

If ENDSCALE is specified in RUNSPEC without the switch 'DIRECT', then none of the directional forms of the SWL keyword are allowed. The SWL keyword then defines the new connate water saturation for computing the water flow across each grid cell face and the transformed capillary pressure table used in the equilibration procedure.

Each SWL keyword should be followed by one real number for each grid block in the current input box specifying the connate water saturation within the grid block. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash(/).
Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If any of the keywords SWL, SWLX, SWLX-, SWLY, SWLY-, SWLZ, SWLZ- are omitted in a model where they could be accepted then the connate water saturation for an oil-water (or gas-water) capillary pressure calculation defaults to the value used in the appropriate water saturation function table.

If a connate water saturation is entered using the SWLPC keyword then this saturation will be used when scaling the capillary pressure curves. The connate water saturation entered using $SWL$ is only used to scale the oil relative permeability curves.

If the Hysteresis model (switch 'HYSTER' in keyword SATOPTS) is active then the $SWL$ data scales the drainage curve. The imbibition curve can be scaled by using the equivalent ISWL (and ISWLX, ISWLX-,...) keyword.

See also keywords SWLPC, SWU and ENPTVD in the PROPS section.

**Note:** In some cases, where messages relating to end-point scaling are emitted, for example, when checking that the scaled end-points are consistent, reference is made to SWCO as an alternative to SWL.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the endpoint scaling procedure.

### Example

```plaintext
-----RUNSPEC section
DIMENS
 1 7 3 /
ENDSCALE
/-----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3
SWL
 5*0.22   10*0.17   3*0.19 /
```
**SWLPC**

### Scaled connate water saturations, for Pc curves only

The SWLPC and ISWLPC keywords scale the connate water end point of the oil-water capillary pressure curves without scaling the corresponding oil relative permeability curves.

The SWLPC keyword should not be used unless water is present in the model (keyword WATER in the RUNSPEC section). The keyword ENDSCALE should be specified in the RUNPSEC section.

The SWLPC keyword should be followed by one real number for each grid block in the current input box, specifying the connate water saturation within the grid block. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If SWLPC is not specified, the oil-water (or gas-water) capillary pressure scaling is performed using the connate water saturation specified with the SWL family of keywords (SWL, SWLX, SWLX-, SWLY, SWLY-, SWLZ, SWLZ-). If in turn any of the keywords SWL, SWLX, SWLX-, SWLY, SWLY-, SWLZ, SWLZ- are omitted, then the connate water saturation for an oil-water (or gas-water) capillary pressure calculation defaults to the value used in the appropriate water saturation function table.

If the hysteresis model (switch 'HYSTER' in keyword SATOPTS) is active then the SWLPC data will scale the drainage curve. The imbibition curve can be scaled by using the equivalent ISWLPC keyword.

See also keywords SWL and ENPTVD in the PROPS section.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the endpoint scaling procedure.

### Example

```plaintext
----RUNSPEC section
DIMENS
  1 7 3 /
ENDSCALE
/
----PROPS section
PROPS
SWLPC
  7*0.22  7*0.20  7*0.18 /
```

**Keywords**

2208
The **SWOF** keyword may be used in runs containing both oil and water as active phases, to input tables of water relative permeability, oil-in-water relative permeability and water-oil capillary pressure as functions of the water saturation. If gas is also an active phase in the run, the gas/oil saturation functions must be input with either keyword **SGOF** or **SLGOF**.

The data comprises **NTSFUN** (see item 1 in keyword **TABDIMS** in the **RUNSPEC** section) tables of water/oil saturation functions, each terminated by a slash (/). Each table consists of 4 columns of data:

1. **The water saturation.**
   - Values should be between 0 and 1 and should increase monotonically down the column.
   - The first value in the column is interpreted as the connate water saturation. The last value in the column is interpreted as $S_w = 1 - S_{or}$.

2. **The corresponding water relative permeability.**
   - Values should be between 0 and 1 and should be level or increasing down the column.
   - The first value in the column must be 0 (zero).

3. **The corresponding oil relative permeability when only oil and water are present.**
   - Values should be between 0 and 1 and should be level or decreasing down the column.
   - If gas is also present in the run, the first value in the column (that is $k_{row}$ at $S_o = S_{omax} = 1 - S_{wco}$) must be the same as $k_{rowg}$ at $S_g = 0$ set in keyword **SGOF** or **SLGOF**. The last value in the column must be zero.

4. **The corresponding water-oil capillary pressure.**
   - Values should be level or decreasing down the column.
   - **UNITS:** bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

Default values (represented by 1*) may be inserted as required in columns 2, 3 and 4. When the table is read in, defaults are replaced by values computed by linear interpolation.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than **NSSFUN** (see item 3 in keyword **TABDIMS**). The entire table may be defaulted provided the table is not the first. Defaulted tables are replaced with a copy of the previous table.

The keyword families

(i) **SWOF**, **SGOF** and **SLGOF**

(ii) **SWFN**, **SGFN**, **SGWFN**, **SOF2** and **SOF3**

provide two alternative formats for inputting the saturation functions. Do not mix keywords from the two families in the same run. In gas-water runs, or if you are using the ECLIPSE 100 miscible flood or solvent options, the saturation functions must be input with keywords from family (ii).
For two dimensional tables the following keywords can also be used: **SOF32D, SGF32D, SWF32D**.

See "Saturation functions" in the *ECLIPSE Technical Description* for more information, and a description of the methods used to compute three phase relative permeabilities.

**Note:** The critical saturation for each relative permeability function is established by determining the highest saturation in the table for which the relative permeability is notionally zero, that is, less than a small threshold value. Relative permeability values which are lower than this threshold value will be treated as zero. For ECLIPSE 100 and ECLIPSE 300 the default threshold values are $1.0 \times 10^{-6}$ and $1.0 \times 10^{-20}$ respectively. These values may be modified using the **TOLCRIT** keyword.

### Example

With **NTSFUN=2** and **NSSFUN ≥8**:

```
   SWOF

   Sw  Krw  Krow  Peow
     .2200   .0000   1.0000    7.0000
     .3000   .0700   0.4000    4.0000
     .4000   .1500   0.1250    3.0000
     .5000  1*       0.0649    1*
     .6000   .3300   0.0048    2.0000
     .8000   .6500   0.0        1.0000
     .9000   .8300   0.0        1*
   1.0000  1.0000   0.0        0.0000 / table 1
     .18   .00     1.00      0
     .32   .07     0.38      0
     .50   .31     0.05      0
     .60   .38     0.004     0
     .80   .57     0.0        0
   1.00   1.00     0.0        0 / table 2
```

**Note:** Slow run times may result if the relative permeability changes abruptly over a small saturation interval, especially if IMPES is used. ECLIPSE honors the data input and does not attempt to overcome convergence problems by retabulating over equally spaced saturation intervals. If chopped relative permeability data is used to minimize dispersion, ECLIPSE performs best in fully-implicit mode if the relative permeability rises steeply from zero (immobility) rather than from a small value.
Scaled saturation table maximum water saturations

The different forms of this keyword specify the maximum water saturation (that is the largest water saturation in a water saturation function table) within each grid cell. The scaled maximum water saturation is used to determine the scaled forms for both oil-water (or gas-water) capillary pressure curves and the water relative permeability curves. The scaled form of the capillary pressure curves is obtained by linearly scaling the tabulated capillary pressure data between the new connate water saturation (see keyword SWL in the PROPS section) and the new maximum water saturation defined using an SWU keyword. A similar linear transformation is used to produce the scaled water relative permeability curves using the new critical water saturation (see keyword SWCR in the PROPS section) and the new maximum water saturation defined using an SWU keyword. The scaled capillary pressure and relative permeability curves are then used to compute the water flow out of each grid cell face, the water flow between grid cells and well connections and in the equilibration algorithm.

Seven keywords are provided to allow the maximum water saturation for each grid cell to be scaled independently. These keywords are:

- **SWU**: Maximum water saturation for the non-directional saturation function tables
- **SWUX**: Maximum water saturation for the +X face of each grid cell
- **SWUX-**: Maximum water saturation for the -X face of each grid cell
- **SWUY**: Maximum water saturation for the +Y face of each grid cell
- **SWUY-**: Maximum water saturation for the -Y face of each grid cell
- **SWUZ**: Maximum water saturation for the +Z face of each grid cell
- **SWUZ-**: Maximum water saturation for the -Z face of each grid cell

In radial grids the same directional keywords should be used to represent the maximum water saturations for flows in the +R, -R, +T, -T, +Z, -Z directions respectively.

None of the SWU keywords should be used if water is not present in the model. The keyword **ENDSCALE** should be specified in the RUNSPEC section.

If both directional and irreversible end-point scaling is requested, using switches 'DIRECT' and 'IRREVERS' in the ENDSCALE keyword in RUNSPEC, then the maximum water saturations for computing the water flow across the +X and -X faces of a grid cell are set using the keywords SWUX, SWUX−. Similarly SWUY, SWUY− are used for the +Y, -Y faces and the keywords SWUZ, SWUZ− for the +Z, -Z faces. Scaling of the oil-water (or gas-water) capillary pressure tables in the equilibration algorithm and scaling of the relative permeability curves to compute flows between grid cells and well connections are performed using the SWU keyword.

When the end point scaling is requested to be directional and reversible, using the switch 'DIRECT' but not 'IRREVERS' in ENDSCALE, the keyword SWUX defines the maximum water saturations for the +X and -X faces of each grid cell. Keywords SWUY, SWUZ are used for both the +Y, -Y faces and the +Z, -Z faces of each grid cell respectively. The maximum water saturation values for use in equilibration procedure and for computing the well connection flows are entered using the SWU keyword.

If **ENDSCALE** is specified in RUNSPEC without the switch 'DIRECT', then none of the directional forms of the SWU keyword are allowed. The SWU keyword then defines the new maximum water saturation for
computing the water flows between adjacent grid cells, well connection flows and in the equilibration algorithm.

Each SWU keyword should be followed by one real number for each grid block in the current input box specifying the maximum water saturation within the grid cell. The saturation should be in the range 0.0 to 1.0 inclusive. The data must be terminated by a slash (/).

Grid blocks are ordered with the X-axis index cycling fastest followed by the Y and Z indices. Repeat counts may be used for repeated values (for example 10*0.21). Note that spaces must not be inserted on either side of the asterisk.

If any of the keywords SWU, SWUX, SWUX–, SWUY, SWUY–, SWUZ, SWUZ– are omitted in a model where they could be accepted then the maximum water saturation for capillary pressure and relative permeability calculations defaults to the value used in the appropriate water saturation function table.

If the Hysteresis model (switch 'HYSTER' in keyword SATOPTS) is active then the SWU data scales the drainage curve. The imbibition curve can be scaled by using the equivalent ISWU (ISWUX, ISWUX–, ...) keyword.

See also keywords SWL, SWCR, ENPTVD in the PROPS section.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the end-point scaling procedure.

Example

```plaintext
-----RUNSPEC section
DIMENS
  1 7 3 /
ENDSCALE
 /
-----PROPS section
-------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
  1 1 2 7 1 3
SWU
18*1.00 /
```

Keywords

2212
This section contains an alphabetic listing of keywords starting with the letter T.
### Table dimensions

The data consists of some or all of the following items which describe the sizes of saturation and PVT tables used in the run, and also the number of fluid-in-place regions. The data must be terminated by a slash (/).

1. **NTSFUN** The number of saturation tables entered using `SGFN` in the `PROPS` section for example. (Different saturation tables may be used in different parts of the reservoir - see `SATNUM` in the `REGIONS` section.)
   
   **DEFAULT:** 1

2. **NTPVT** The number of PVT tables entered using `PVTG` and `PVTO` in the `PROPS` section for example. (Different PVT tables may be used in different parts of the reservoir - see `PVNUM` in the `REGIONS` section.)
   
   **DEFAULT:** 1

3. **NSSFUN** The maximum number of saturation nodes in any saturation table - for example `SGFN` and `SGOF` in the `PROPS` section.
   
   **ECLIPSE 100**
   **DEFAULT:** 20

   **ECLIPSE 300**
   **DEFAULT:** 50

4. **NPVPT** The maximum number of pressure nodes in any PVT table (`PVTG`, `PVTO`, `PVDG` and `PVDO` for example), or in any rock compaction table (`ROCKTAB`).
   
   **ECLIPSE 100**
   **DEFAULT:** 20

   **ECLIPSE 300**
   **DEFAULT:** 50

5. **NTFIP** The maximum number of FIP regions defined using `FIPNUM` in the `REGIONS` section.
   
   **DEFAULT:** 1

   This may also be set using item 1 of the `REGDIMS` keyword. If it is set in both places, the greater of the two settings is used. Additionally, if item 1 of `DYNRDIMS` is defined, the greatest of the three settings is used.

6. **NRPVT**
   
   The maximum number of R<sub>v</sub> nodes in a live oil PVT table (`PVTO` or `PVCO`) or R<sub>r</sub> nodes in a wet gas PVT table (`PVTG`). This item is ignored unless either `DISGAS` or `VAPOIL` is specified in the `RUNSPEC` section. In API tracking runs this number needs to be the maximum number of distinct valued R<sub>v</sub> nodes across all the PVT tables.

   **ECLIPSE 300 only**
   The maximum number of R<sub>v</sub> nodes in a live oil PVT table (`PVTO` or `PVCO`).
   
   **DEFAULT:** 20

7. **NTENDP** The number of saturation end-point versus depth tables is specified.
   
   This item may also be specified by item 3 of keyword `ENDSCALE`; if set in both places the maximum of the two values will be used.
9. The number of equation of state regions for reservoir conditions. The maximum value which can be specified in this item is 10.
   DEFAULT: 1

10. The number of equation of state regions for surface conditions. The maximum value which can be specified in this item is 10.
    DEFAULT: Value in item 9.

11. The maximum number of flux regions.
    This item may also be specified by item 4 of keyword REGDIMS; if set in both places the maximum of the two values will be used.
    DEFAULT: 10

12. The maximum number of thermal regions.
    DEFAULT: 1

13. NTROCC The number of rock tables (see keyword ROCK).

14. The maximum number of pressure maintenance regions (see keyword GPMAINT).
    DEFAULT: 0

15. The maximum number of temperature dependent K-value tables.
    (Thermal option only)
    DEFAULT: 0
    A maximum of nine temperature dependent K-value tables are allowed.

16. NTALPHA The number of transport coefficient table (ALPHA, ALPHAD, ALPHAI) regions
    DEFAULT: NTPVT

17. Asphaltene: the maximum number of rows in the ASPKDAM keyword table when permeability damage is not correlated to porosity change.
    DEFAULT: 10

18. Asphaltene: the maximum number of rows in the ASPREWG keyword table.
    DEFAULT: 10

19. Asphaltene: the maximum number of rows in the ASPVISO table when a look-table is used to model the oil viscosity changes with asphaltene precipitate

Note: If the dataset contains no other rock-related keywords, this value must be set explicitly to a value greater than zero in order to allow a dataset which contains the ROCKNUM keyword to be read. If it is defaulted, PVTNUM will be used instead.
DEFAULT: 10

20. Not used

ECLIPSE 300 only

21. Asphaltene: the maximum number of columns in the ASPPW2D keyword table
   DEFAULT: 5

22. Asphaltene: the maximum number of rows in the ASPPW2D keyword table
   DEFAULT: 5

ECLIPSE 300 only

23. Asphaltene: the maximum number of rows in the ASPWETF keyword table
   DEFAULT: 5

ECLIPSE 300 only

24. The number of composition dependent K-value tables.
   DEFAULT: 0
   A maximum of nine composition dependent K-value tables are allowed.

ECLIPSE 100 only

25. Reserved.

**Example**

Two saturation and pressure tables with 50 nodes in each table.

```
TABDIMS
  2  2  50  50 /
```
**TBASIS**

**Specify basis for tensor permeabilities**

This keyword specifies the basis for the permeability tensor. If tensor permeabilities are entered then the discretization will account for the tensor permeabilities in a consistent manner.

The keyword should be followed by a single item followed by a slash (/).

1. Specify the basis in which the permeability tensor is defined. The permeability tensor \( K \) is assumed to be diagonal and is specified by its physical components \( K_{11} = \text{PERMX}, \ K_{22} = \text{PERMY}, \ K_{33} = \text{PERMZ}, \ K_{12} = \text{PERMXY}, \ K_{23} = \text{PERMYZ} \) and \( K_{31} = \text{PERMZX} \).

   The tensor may be specified in one of the following bases:

   1. \( K \) is defined in the Cartesian x, y and z axes.

   2. \( K \) is defined in each corner point cell relative to the coordinate system formed by joining the midpoints of opposite faces in a corner point cell.

   3. \( K \) is defined in the bedding plane by the vectors obtained by joining the midpoints of opposite faces in the corner point cell in the logical \( i \) and \( j \) directions. The third basis vector is chosen normal to the bedding plane.

   DEFAULT: 2

See "Tensor permeability" in the *ECLIPSE Technical Description* for further information.

**Example**

```
TBASIS
1 /
```
**Initial tracer concentrations**

The keyword specifies the initial concentration of a tracer in each grid block. The actual keyword to be used is a concatenated name of up to 8 characters in length, which consists of the following segments:

**ECLIPSE 100**

Segment:

1. Characters 1-4 must be the character string TBLK
2. Character 5 must be the letter F or S

For a tracer whose associated stock tank phase can only exist in the free state, the letter F (free) must be used. This case includes all water phase tracers, oil phase tracers unless VAPOIL is specified in the RUNSPEC section, and gas phase tracers unless DISGAS is specified in RUNSPEC.

For a tracer whose associated stock tank phase can exist in the solution state (for example, gas when DISGAS is specified in the RUNSPEC section, and oil when VAPOIL is in RUNSPEC) the letter S (solution) is allowed. A tracer that is embedded in such a phase requires both the free and solution component concentrations to be initialized.

3. Characters 6-8 must be the name of the tracer which is being initialized.

If a unit for tracer amount has been supplied in item 3 of keyword TRACER, then the unit of tracer concentration is

**UNITS:** TR/sm³ (METRIC), TR/stb for liquid phase tracers, TR/Mscf for gas phase tracers (FIELD), TR/scc (LAB)

where TR is the unit supplied for the tracer.

**ECLIPSE 300**

Segment:

1. Characters 1-4 must be the character string TBLK
2. Characters 5-8 must be the name of the tracer which is being initialized.

The keyword should be followed by one real number for every grid block specifying the initial tracer concentration. The data field should be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS.

Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*).

**Notes**

Values can also be set using the full range of data manipulation keywords such as **EQUALS**, **MULTIREG** or **OPERATE**.
Examples

Example 1

ECLIPSE 100

Where DISGAS is specified; NDIVIX=8, NDIVIY=6 and NDIVIZ=3 are specified in RUNSPEC keyword DIMENS; and the gas phase tracer is SGS:

```
TBLKFSGS
  48*0.0
  48*1.0
  48*1.0  /
TBLKSGS
  48*0.0
  48*0.0
  48*0.0  /  
```

Example 2

Where NDIVIX=2, NDIVIY=3 and NDIVIZ=3 are specified in RUNSPEC keyword DIMENS and the water phase tracer is WT1:

```
TBLKFWT1
  18*1.0  /  
```

Example 3

ECLIPSE 300

```
TBLKTRAA
  48*0.0
  48*1.0
  48*1.0  /  
```
**TBOIL**

**Component boiling points**

In a run with \( N_c \) components, and which uses the Zudkevitch Joffe equation, this keyword specifies the pure component boiling points. The keyword should be followed by \( N_c \) values.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

UNITs: K (METRIC), °R (FIELD), K (LAB), K (PVT-M)

**Example**

```
TBOIL
350.46
450.34
```
Boiling points for surface EoS

This optional keyword supplies boiling points for use in the surface calculations. The default, if this is not supplied, is for these to follow the reservoir values supplied with keyword TBOIL.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.
**Temperature cutback table dimensions**

This keyword can only be used with the THERMAL option (see "Thermal option" in the ECLIPSE Technical Description).

The data consists of the following items, describing the dimensions of the TCB table data to be used in the run.

1. The number of temperature cutback tables.
   - DEFAULT: 1

2. The maximum number of rows in any TCB table.
   - DEFAULT: 10

The use and interpretation of temperature cutback tables is discussed in the definition of the keyword WELLTCBT.
Critical temperatures

In a run with $N_c$ components, this keyword associates a critical temperature with each component.

The keyword should be followed by $N_c$ values.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

UNITs: K (METRIC), °R (FIELD), K (LAB), K (PVT-M)

DEFAULT: There are no component-wise default values. The users are recommended not to default these to the internal default value.

Example

```
TCRIT 140 270 450 670 /
```

Note:
For live oil properties - a component can be made nonvolatile or non-condensable using the CVTYPE keyword. This keyword will also reset the component critical temperature, to $10^4$ for nonvolatile components and to $10^-6$ for non-condensable components. For back-compatibility with earlier versions of ECLIPSE, components with critical temperatures of $10^4$ and above will also be assumed to be nonvolatile, while those with temperatures of $10^-6$ will be assumed to be non-condensable. For surface calculations, the volatility types are set by default to the reservoir values. If different volatility types are required at the surface, the CVTYPES keyword can be used. In this case the TCRITS keyword should also be used to ensure the correct critical temperatures are used in the surface calculations.

The Thermal option is described in “Thermal option” in the ECLIPSE Technical Description.
Critical temperatures for detailed components

In a lumped run with \( N_d \) detailed components, this keyword associates a critical temperature with each of the detailed components. Lumped simulations are enabled with the LUMPDIMS keyword in the RUNSPEC section; further details can be found in "Delumping and lumping" in the ECLIPSE Technical Description.

The keyword should be followed by a record of \( N_d \) values, terminated with a slash, for each reservoir equation of state region. The values represent the critical temperature of each detailed component, in the order in which the detailed components are defined in the LUMPING keyword.

If multiple equations of state are being used, a record of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

UNITs: K (METRIC), °R (FIELD), K (LAB), K (PVT-M)

Example

```
TCRITDET
140 270 450 670 /
```
Critical temperatures for surface EoS

This optional keyword supplies critical temperature values for use in the surface calculations. The default, if this is not supplied, is for these to follow the reservoir values supplied with keyword TCRIT.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.
TCRITSDE Critical temperatures for surface EoS for detailed components

This optional keyword supplies critical temperature values for use in the surface calculations of a lumped simulation. The default, if this is not supplied, is for these to follow the reservoir values supplied with keyword TCRITDET. Lumped simulations are enabled with the LUMPDIMS keyword in the RUNSPEC section; further details can be found in "Delumping and lumping" in the ECLIPSE Technical Description.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.
**TECBDI**

### Initial tectonic boundary conditions

This keyword specifies initial tectonic boundary conditions when initial stress calculation INIT is specified with the keyword GEOMECH in the RUNSPEC section.

Use of this keyword allows initial tectonic loadings to be specified on side boundaries (X-, X+, Y-, and Y +). A tectonic loading at side face is a force/unit area. If a particular boundary is not specified, then that boundary is defaulted to a rigid boundary.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the velocity boundary condition is applied
   
   This must be one of the following:
   
   - X+ Specify the boundary condition on the outer X+ edge
   - X- Specify the boundary condition on the outer X- edge
   - Y+ Specify the boundary condition on the outer Y+ edge
   - Y- Specify the boundary condition on the outer Y- edge
   
   DEFAULT: There is no default for this item; it must be specified.

2. A tectonic loading which is a normal traction on this boundary.
   
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   
   DEFAULT: 0

3. First block to be modified in the X direction
   
   DEFAULT: 1 if an X- face

4. Last block to be modified in the X direction
   
   DEFAULT: NX if an X+ face

5. First block to be modified in the Y direction
   
   DEFAULT: 1 if a Y- face

6. Last block to be modified in the Y direction
   
   DEFAULT: NY if a Y+ face

7. First block to be modified in the Z direction
   
   DEFAULT: 1 if a Z- face

8. Last block to be modified in the Z direction
   
   DEFAULT: NZ if a Z+ face
Example

```
-- Set a 3000 psi total compressive traction on the X+, X-, Y+ and Y- boundaries
TECDI
  X+  -3000 /
  X-  -3000 /
  Y+  -3000 /
  Y-  -3000 /
/```

Keywords

2228
TECBDIL

Initial tectonic boundary conditions in an LGR

This keyword specifies initial tectonic boundary conditions in an LGR when initial stress calculation INIT is specified with the keyword GEOMECH in the RUNSPEC section.

This keyword allows initial tectonic loadings to be specified on side boundaries (X-, X+, Y-, and Y+) in a local grid refinement. A tectonic loading at side face is a force/unit area. If a particular boundary is not specified, then that boundary is defaulted to a rigid boundary.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the velocity boundary condition is applied
   This must be one of the following:
   X+ Specify the boundary condition on the outer X+ edge
   X– Specify the boundary condition on the outer X- edge
   Y+ Specify the boundary condition on the outer Y+ edge
   Y– Specify the boundary condition on the outer Y- edge
   DEFAULT: There is no default for this item; it must be specified.

2. Name of the local grid refinement.

3. A tectonic loading which is a normal traction on this boundary.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 0

4. First block to be modified in the X direction
   DEFAULT: 1 if an X- face

5. Last block to be modified in the X direction
   DEFAULT: NX if an X+ face

6. First block to be modified in the Y direction
   DEFAULT: 1 if a Y- face

7. Last block to be modified in the Y direction
   DEFAULT: NY if a Y+ face

8. First block to be modified in the Z direction
   DEFAULT: 1 if a Z- face

9. Last block to be modified in the Z direction

Keywords
DEFAULT: NZ if a Z+ face

Example

```plaintext
-- For the local grid refinement LGR1 set a 3000 psi total compressive traction on
the X+, X-, Y+ and Y- boundaries

TECBDIL
X+ 'LGR1' -3000 /
X- 'LGR1' -3000 /
Y+ 'LGR1' -3000 /
Y- 'LGR1' -3000 /
/```

Keywords

2230
The temperature option is required

This indicates that the Temperature option is required, to allow the modeling of the temperature effects of cold water injection. See "Temperature option" in the ECLIPSE Technical Description for more information.

If the dual porosity option is also used, see the thermal option documentation for "Dual porosity" in the ECLIPSE Technical Description for how the rock volume is represented.

The keyword has no associated data.
The initial temperature can also be set using the TEMPVD or RTEMP keyword, or in Thermal simulations by using the GETSOL keyword to read TEMP data from a Restart file.

This keyword cannot be used with the EQUIL keyword. Instead, temperature data should be entered using either the TEMPVD keyword or RTEMP keyword.

The keyword should be followed by \( N_x \times N_y \times N_z \) values in normal order, with the X index changing fastest.

UNITs: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

This keyword can also be used with the EQUALS keyword.

**Example**

```
TEMPI
100*240 100*250 100*260 100*270 100*280 /
```
**TEMPNODE**  Temperature nodes for polymer solution viscosity

This keyword describes the nodal values for temperature, to be used in the calculation of the polymer solution viscosity when the temperature option (TEMP) is active. The data comprises NTPVT (see RUNSPEC keyword TABDIMS) tables of temperature values, each terminated by a slash (/).

Each table consists of up to NPPVT (see RUNSPEC keyword TABDIMS) values of the temperature. Within each table the values should be positive and strictly increasing.

**UNITS:** °C (METRIC), °F (FIELD), °C (LAB)

The number of entries in each table must correspond to the number of polymer solution viscosity entries used in the PLYVISCT keyword or to the number of polymer-solution/brine viscosity entries used in the PLYVSCST keyword.

**Example**

With NTPVT=2 and NPPVT ≥ 4:

```
TEMPNODE
10
50
100
150 /
10
150 /
```
**TEMPTVD**  
Requests flux limited transport for the temperature option

This keyword requests the use of the flux limited transport scheme for the temperature option. Flux limited transport is a higher-order method that reduces numerical dispersion of the energy and hence the temperature.

The option is not available in cases with local grid refinement; in these cases the keyword is ignored.

The method is analogous to the flux limited scheme for single phase tracers initiated with the TRACTVD keyword.

See "Control of numerical diffusion" in the ECLIPSE Technical Description for further information.

The TEMPTVD keyword has no associated data.

**Example**

```
TEMPTVD
```

---

**Keywords**

2234
Temperature versus depth data

The data comprises a table of temperature versus depth data for each equilibration region. Each table consists of 2 columns of data, and is terminated with a slash (/).

The maximum number of rows in the table is set in item 3 in keyword EQLDIMS.

Column:

1. Depth values.
   The values should increase monotonically down the column.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

2. The corresponding temperature values.
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

For depth values outside the interval specified, constant value extrapolation is performed.

The temperature value is obtained for each cell, and this is used to evaluate equation of state expressions during the run. Note that the temperature does not change - that is, heat is not transported around the reservoir by fluid flows. All fluid properties, such as density and viscosity, use the cell temperatures. There is a small extra CPU cost in using the TEMPVD option.

In thermal simulations this keyword can be used to set the initial temperature in the reservoir.

Temperatures can also be set using the RTEMP keyword, or if an explicit initialization is used, the TEMPI keyword can be used.

Example

```
TEMPVD
4000 220
6000 245
/
```
THANALB  Use analytic water densities

This keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section).

THANALB requests that analytic water and steam densities be used instead of being looked up from internal tables. The analytic expression used is described in section “Water density” in the ECLIPSE Technical Description.

This keyword is set by default.
**THANALH**  

Use analytic water enthalpies

This keyword can only be used with the THERMAL option (keyword **THERMAL** in the RUNSPEC section).

**THANALH** requests that analytic water and steam enthalpies be used instead of being looked up from internal tables. The analytic expression used is described in the section “Energies and enthalpies” in the *ECLIPSE Technical Description*. 

---

**Keywords**

- ECLIPSE 100
- ECLIPSE 300
- SPECIAL
  - RUNSPEC
  - GRID
  - EDIT
- PROPS
- REGIONS
- SOLUTION
- SUMMARY
- SCHEDULE
THANALV  Use analytic oil viscosities

This keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section).

THANALV requests that analytic oil viscosities be used, as described in the section "Viscosity" in the ECLIPSE Technical Description. In LIVEOIL runs, analytic formulae can also be specified for each component using the OILVISCC or OILVISCF keywords. Alternatively oil viscosities can be looked up from the OILVISCT table in both DEADOIL and LIVEOIL runs.

If both the PVCO and VISCREF keywords are specified, a pressure dependence modifies the above temperature dependent viscosities. The oil viscosity at the prevailing pressure is then calculated as follows:

\[
\mu_o = \mu(T) \frac{\mu(P)}{\mu(P_{ref})}
\]

Eq. 3.203

where

\(\mu(T)\)  Viscosity from the THANALV keyword.

\(\mu(P)\)  Viscosity from the PVCO keyword at the prevailing pressure.

\(\mu(P_{ref})\)  Viscosity as entered by the PVCO keyword at the reference pressure as defined by the VISCREF keyword.

This keyword contains no items.
**THCGAS**

**Gas phase thermal conductivity**

This keyword may only be used when the ECLIPSE 300 Thermal option (keyword `THERMAL` in the RUNSPEC section) is active.

This keyword should be used with the `THCROCK` keyword.

The `THCGAS` keyword defines the thermal conductivities of the gas phase. This will be used to determine a porosity weighted average of the phase and rock conductivities (THCAVE), which will then be used to calculate the thermal conduction of heat in the reservoir.

\[
THCAVE = \phi \left( \frac{THCOIL + THCGAS + THCWATER + THCSOLID}{NPHAS} \right) + (1-\phi)THCROCK
\]

In the expression above \( \phi \) is the grid porosity and NPHAS denotes the number of phases in the model. A heat conduction multiplier is then calculated and applied at each timestep to take into account the fluid saturations.

See "Thermal option" in the *ECLIPSE Technical Description* for details of this option.

Refer also to keywords `THCOIL`, `THCWATER`, and `THCSOLID`.

The keyword should be followed by one non-negative real number for every grid block in the current input box, specifying the gas phase thermal conductivities. The data must be terminated by a slash (/).

**UNITS:** kJ/m/day/K (METRIC), Btu/ft/day/^°R (FIELD), J/cm/hr/K (LAB), kJ/m/day/K (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

**Example**

```plaintext
THCGAS
1200*25.0 /
```
**THCOIL**

**Oil phase thermal conductivity**

This keyword may only be used when the ECLIPSE 300 Thermal option (keyword `THERMAL` in the `RUNSPEC` section) is active.

This keyword should be used with the `THCROCK` keyword.

The `THCOIL` keyword defines the thermal conductivities of the oil phase. This will be used to determine a porosity weighted average of the phase and rock conductivities (THCAVE), which will then be used to calculate the thermal conduction of heat in the reservoir.

\[
\text{THCAVE} = \phi \left( \frac{\text{THCOIL} + \text{THCGAS} + \text{THCWATER} + \text{THCSOLID}}{\text{NPHAS}} \right) + (1-\phi)\text{THCROCK}
\]

In the expression above \( \phi \) is the grid porosity and NPHAS denotes the number of phases in the model. A heat conduction multiplier is then calculated and applied at each timestep to take into account the fluid saturations.

See "Thermal option" in the ECLIPSE Technical Description for details of this option.

Refer also to keywords `THCGAS`, `THCWATER`, and `THCSOLID`.

The keyword should be followed by one non-negative real number for every grid block in the current input box, specifying the oil phase thermal conductivities. The data must be terminated by a slash (/).

**UNITS:** kJ/m/day/K (METRIC), Btu/ft/day/°R (FIELD), J/cm/hr/K (LAB), kJ/m/day/K (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

**Example**

```
THCOIL
1200*25.0 /
```
**Matrix-fracture conductivity**

This keyword specifies the matrix to fracture thermal conductivity value for each matrix cell in a dual-porosity reservoir, for the THERMAL option (keyword THERMAL in the RUNSPEC section).

The keyword should be followed by \((N_x N_y N_z)^/2\) values in normal order, with the X index changing fastest.

**UNITS:** kJ/m/day/K (METRIC), Btu/ft/day/°R (FIELD), J/cm/hr/K (LAB), kJ/m/day/K (PVT-M)

The Dual Porosity and Dual Permeability options are discussed in more detail in “Dual porosity” of the ECLIPSE Technical Description. The Thermal option is described in “Thermal option” in the ECLIPSE Technical Description.

**Example**

Set conductivities for a 10*10 cross section,

```
THCONMF
 100*3 /
```
**THCONR**

### Thermal conductivity of rock and fluids

This keyword may only be used when either the ECLIPSE 100 Temperature option (keyword TEMP in the RUNSPEC section) or the ECLIPSE 300 Thermal option (keyword THERMAL in the RUNSPEC section) is active.

The THCONR keyword supplies the combined rock and fluid thermal conductivities as a porosity weighted average of the phase and rock conductivities (THCAVE), which will then be used to calculate the thermal conduction of heat in the reservoir.

See "Temperature option" and "Thermal option" in the ECLIPSE Technical Description for details of these options.

For the ECLIPSE 300 Thermal Option only, this keyword **must not** be used with either THCROCK, THCOIL, THCGAS, THCWATER or THCSOLID.

The keyword should be followed by one non-negative real number for every grid block in the current input box, specifying the combined rock and fluid thermal conductivities. The data must be terminated by a slash (/).

**UNITS:** kJ/m/day/K (METRIC), Btu/ft/day/°R (FIELD), J/cm/hr/K (LAB), kJ/m/day/K (PVT–M)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

**Note:** For the ECLIPSE 300 Thermal Option only, the thermal conductivity of the reservoir can also be dependent on the gas saturation via the THCONSF keyword.

### Example

```
THCONR
1200*25.0 /
```
Saturation-dependence of conductivity

This keyword may be used if the THERMAL option is active (keyword THERMAL in the RUNSPEC section).

This keyword can only be used with the thermal conductivities specified in THCONR. This keyword **must not** be used with THCROCK, THCOIL, THCGAS, THCWATER or THCSOLID.

The THCONSF keyword is used to set up a multiplier, \( \Lambda \), which is a function of \( S_g \). Using this keyword, it is assumed that the thermal conductivity of a rock filled with liquid is independent of whether the liquid is oil or water and is the same whatever the porosity of the rock:

\[
\Lambda = \Lambda (S_g), \quad \Lambda = (1-\alpha S_g)
\]

Eq. 3.204

with \( \alpha \) in the range 0 to 1. This keyword specifies the value of \( \alpha \).

Thermal transmissibilities are calculated in a similar manner to the flow transmissibilities with the thermal conductivities being used instead of the permeabilities. These thermal transmissibilities can also be multiplied by a saturation-dependent factor, \( \Lambda \), as described above.

The keyword should be followed by \( N_xN_yN_z \) values in normal order, with the X index changing fastest.

The thermal option is described in "Thermal option" in the ECLIPSE Technical Description.

Example

Set values for a 10*10 cross section,

```
THCONSF
100*0.1 /
```

Example
**THCROCK**

**Thermal conductivity of rock**

This keyword may only be used when the ECLIPSE 300 Thermal option (keyword THERMAL in the RUNSPEC section) is active.

This keyword must not be used with the THCONR keyword.

The THCROCK keyword defines the thermal conductivities of the rock. This will be used to determine a porosity weighted average of the phase and rock conductivities (THCAVE), which will then be used to calculate the thermal conduction of heat in the reservoir.

\[
THCAVE = \phi \left( \frac{THCOIL + THCGAS + THCWATER + THCSOLID}{NPHAS} \right) + (1-\phi)THCROCK
\]

In the above expression is the \( \phi \) grid porosity and \( NPHAS \) denotes the number of phases in the model. A heat conduction multiplier is then calculated and applied at each timestep to take into account the fluid saturations.

See "Thermal option" in the ECLIPSE Technical Description for details of this option.

Refer also to keywords THCOIL, THCGAS, THCWATER and THCSOLID.

The keyword should be followed by one non-negative real number for every grid block in the current input box specifying the rock thermal conductivities. The data must be terminated by a slash (/).

**UNITS:** kJ/m/day/K (METRIC), Btu/ft/day/°R (FIELD), J/cm/hr/K (LAB), kJ/m/day/K (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

**Example**

```
THCROCK
1200*25.0 /
```

---

**Keywords**

2244
**THCSOLID**

**Solid phase thermal conductivity**

This keyword may only be used when the ECLIPSE 300 Thermal option (keyword `THERMAL` in the RUNSPEC section) is active.

This keyword should be used with the `THCROCK` keyword.

The `THCSOLID` keyword defines the thermal conductivities of the solid phase. This will be used to determine a porosity weighted average of the phase and rock conductivities (`THCAVE`), which will then be used to calculate the thermal conduction of heat in the reservoir.

\[
THCAVE = \varphi \left( \frac{THCOIL + THCGAS + THCWATER + THCSOLID}{NPHAS} \right) + (1-\varphi)THCROCK
\]

In the expression above `\varphi` is the grid porosity and `NPHAS` denotes the number of phases in the model. A heat conduction multiplier is then calculated and applied at each timestep to take into account the fluid saturations.

See "Thermal option" in the *ECLIPSE Technical Description* for details of this option.

Refer also to keywords `THCOIL`, `THCGAS` and `THCWATER`.

The keyword should be followed by one non-negative real number for every grid block in the current input box, specifying the solid phase thermal conductivities. The data must be terminated by a slash (/).

**UNITS:** kJ/m/day/K (METRIC), Btu/ft/day/°R (FIELD), J/cm/hr/K (LAB), kJ/m/day/K (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

**Example**

```
THCSOLID
1200*25.0 /
```
THCWATER  Water phase thermal conductivity

This keyword may only be used when the ECLIPSE 300 Thermal option (keyword THERMAL in the RUNSPEC section) is active.

This keyword should be used with the THROCK keyword.

The THCWATER keyword defines the thermal conductivities of the water phase. This will be used to determine a porosity weighted average of the phase and rock conductivities (THCAVE), which will then be used to calculate the thermal conduction of heat in the reservoir.

\[
THCAVE = \phi \frac{(THCOIL + THCGAS + THCWATER + THCSOLID)}{NPHAS} + (1 - \phi)THROCK
\]

In the expression above \(\phi\) is the grid porosity and NPHAS denotes the number of phases in the model. A heat conduction multiplier is then calculated and applied at each timestep to take into account the fluid saturations.

See "Thermal option" in the ECLIPSE Technical Description for details of this option.

Refer also to keywords THCOIL, THCGAS and THCSOLID.

The keyword should be followed by one non-negative real number for every grid block in the current input box, specifying water phase thermal conductivities. The data must be terminated by a slash (/).

UNITS: kJ/m/day/K (METRIC), Btu/ft/day/°R (FIELD), J/cm/hr/K (LAB), kJ/m/day/K (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Note that spaces must not be inserted on either side of the asterisk.

Example

```
THCWATER
1200*25.0 /
```
Request thermal option

This keyword enables the THERMAL option (see "Thermal option" in the ECLIPSE Technical Description).

Either the DEADOIL or LIVEOIL keyword must also be specified. LIVEOIL is the default.

If the dual porosity option is also used, see the thermal option documentation for "Dual porosity" in the ECLIPSE Technical Description for how the rock volume is represented.

This keyword has no associated data.

Note: A list of features that are not compatible with the thermal option is given in the note at the end of "Instability in AIM runs due to thermal conduction" in the ECLIPSE Technical Description.
**Thermal expansion coefficient**

This keyword specifies the first thermal expansion coefficient for each of the \( N_c \) liquid hydrocarbon components.

The expression used to calculate oil component liquid densities is the following:

\[
\rho_c = \frac{\rho_{ref_c}}{\left[1 + c c P c \left(P - P_{ref_c}\right) \left(T - T_{ref_c}\right)\right]}
\]

Eq. 3.205

where

- \( \rho_{ref_c} \) is the reference density, specified with keyword \texttt{DREF},
- \( c c P c \) is the oil component liquid compressibility, specified with keyword \texttt{CREF},
- \( P_{ref_c} \) is the reference pressure, specified with keyword \texttt{PREF},
- \( T_{ref_c} \) is the reference temperature, specified with keyword \texttt{TREF},

and \( c T_1 c \) is the thermal expansion coefficient, specified with this keyword.

There should be one value for each component:

**UNITS:** 1/K (\texttt{METRIC}), 1/°R (\texttt{FIELD}), 1/K (\texttt{LAB}), 1/K (\texttt{PVT-M})

**DEFAULT:** 0.0

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of \texttt{TABDIMS}. The use of multiple EoS regions is discussed in "Equations of state regions" in the \texttt{ECLIPSE Technical Description}.

**Example**

```
THERMEX1
-- thermal expansion coefficients for a system with 3 hydrocarbon components
3.5E-06  3.5E-06  4.5E-06  /
```
THERMNUM  Thermal region numbers

The THERMNUM keyword should be followed by one integer for every grid block in the current input box specifying the thermal region to which it belongs. The region number should not be less than 0 or greater than the number of thermal regions, specified with TABDIMS item 12.

For DEADOIL runs, the thermal region number should be set to 0 for those grid blocks that have a constant temperature throughout the run. The temperature is set to the initial temperature, and the thermal conductivity of the block is set to 0. This option is not available for LIVEOIL simulations.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example, 115*2). Note that spaces must not be inserted on either side of the asterisk.

Example

```
THERMNUM
144*1 96*0 /
```
Heat transmissibility factor for flux boundary option

This keyword should only be used in a Thermal USEFLUX run, and only if the base full field model was an ECLIPSE 300 Blackoil model or a Compositional model.

For any type of simulation, the transmissibilities are taken from the base run and stored in the flux file.

For a Thermal simulation the heat transmissibilities are needed. However, these are not calculated from a full field non-thermal simulation.

So, for a USEFLUX case where the base run is non-thermal, and the reduced run is thermal, the heat transmissibilities are calculated as follows:

\[ \Psi_{ni} = \alpha \times \Upsilon_{ni} \]

where

\[ \Psi_{ni} \] is the heat transmissibility between cells \( n \) and \( i \)

\[ \Upsilon_{ni} \] is the flow transmissibility between cells \( n \) and \( i \)

and \( \alpha \) is the heat transmissibility factor, defined using this keyword.

DEFAULT: 0.0

The flux boundary option is discussed in more details in "Flux boundary conditions" in the ECLIPSE Technical Description.

**Example**

```
THFLXTF
  0.12 /
```

**Keywords**

- **2250**
Threshold pressures

This keyword sets the threshold pressures for flow between adjacent equilibration regions. The threshold pressure switch THPRES must be set in keyword EQLOPTS in the RUNSPEC section. The facility prevents flow between adjacent equilibration regions until the potential difference exceeds the specified threshold value. Thereafter the potential difference for flow across equilibration region boundaries is reduced by the appropriate threshold value.

When the Threshold Pressure option is active (THPRES specified in keyword EQLOPTS), each interface between adjacent equilibration regions has its own threshold pressure. In fact, if the threshold pressures are irreversible (that is, if the switch IRREVERS is also specified in keyword EQLOPTS), each interface has two threshold pressures, one for flow in each direction.

The threshold pressure is set to zero for any interface not specified with the THPRES keyword. If the threshold pressures are irreversible, the threshold pressure should be specified separately for each direction; otherwise it is set to zero for the direction not specified.

Threshold pressures can be output to the Print file using the argument THPRES in keyword RPTSOL.

The THPRES keyword should be followed by a number of data records, terminated with a line containing just a slash (/). Each record consists of three numbers, and is terminated with a slash. The numbers represent:

1. From equilibration region number (I)
   Must not exceed NTEQUL (see keyword EQLDIMS).
2. To equilibration region number (J)
   Must not exceed NTEQUL.
3. Threshold pressure for flow from region I to region J.

   If this quantity is defaulted (by terminating the record after item 2), the threshold pressure for this interface (and this direction if the threshold pressures are irreversible) will be calculated by the program from the initial conditions. It is set equal to the maximum potential difference between communicating cells across the interface, for any phase that is present in the upstream cell. (Thus, for example, the oil potential difference in the water zone does not influence the threshold pressure.)

   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

End the record with a slash (/).

Terminate the set of records with a line containing just a slash.

See also the THPRESFT keyword.

Example

NTEQUL=3, irreversible threshold pressures:

```
-- RUNSPEC section
EQLDIMS
  3 /
EQLOPTS
THPRES IRREVERS /
-- SOLUTION section
THPRES
  1 2 12.0 /
  2 1  5.0 /
  1 3  7.0 /
```

Keywords
Since individual threshold pressure values have been set by the user, in either direction, between regions 1 and 2, and also regions 1 and 3, these values will be used. If there is communication between regions 2 and 3, threshold pressure values will be calculated from the conditions at the start of the run; otherwise they will be set to zero. The same principle will apply for communication between regions 3 and 2. Threshold pressures can be output to the Print file using the argument \texttt{THPRES} in keyword \texttt{RPTSOL}.
THPRESFT

Sets up a threshold pressure for a fault

The THPRESFT keyword can be used to set up a threshold pressure for a fault previously defined using the FAULTS keyword. The threshold pressure switch THPRES must be set in keyword EQLOPTS in the RUNSPEC section. The facility prevents flow occurring between adjacent grid blocks either side of the fault until the potential difference exceeds the specified threshold value. Thereafter the potential difference for flow across the fault is reduced by the appropriate threshold value.

The keyword is followed by any number of data records, terminated with a blank record containing just a slash (/). Each record contains the following two items of data, and is terminated with a slash (/).

1. Fault name (as specified in the FAULTS keyword), or a fault name root.
   A fault name root, ending with an asterisk (*), can be used to refer to several faults in one record.

2. Threshold pressure.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
   DEFAULT: 0.0

The set of records must end with a blank record containing only a slash (/).

See also the keywords FAULTS and THPRES.

Notes

- If a fault is referred to twice in one or more THPRESFT keywords, the threshold pressure is taken from the last entry.

- The fault threshold pressure can be used in combination with the equilibration region based threshold pressure option (keyword THPRES). If two threshold pressures are applied to the flow between the same two grid blocks, the acting threshold pressure is taken as the maximum of the two threshold pressures.

- Threshold pressures cannot be set for a fault which crosses a local grid or coarsened region.

Example

See the FAULTS keyword example.

```
THPRESFT
   -- Threshold pressure
   'zigzag'  10.0  /
   'block'   1.2   /
```

Keywords

2253
Use IAPWS 97 steam table

This keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section).

If the keyword THSTT97 is used, the properties of water and steam will be based on an analytic formulation, as detailed by *The International Association for the Properties of Water and Steam* in the *IAPWS Industrial formulation 1997* (IAPWS-IF97). The water and steam densities, the saturation pressures and temperatures, and the water and steam energies and enthalpies will be calculated using the IAPWS-IF97 formulation.
Steam viscosity coefficients

This keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section).

THSVC allows the user to specify the coefficients $A_g$, $B_g$, $C_g$ and $D_g$ in the expression for the steam viscosity

$$\mu_g = A_g + B_g T_C + C_g P_p D_g$$

where the temperature $T_C$ is in °C and the pressure $P_p$ is in MPa. The default values are chosen to match the steam viscosity over a large temperature range. However, if the simulation occurs over a relatively narrow temperature range the coefficients may be modified to improve this fit.

**Note:** Regardless of the unit system used for the simulation the steam viscosity expression is always evaluated with the temperature in °C and pressure in MPa.

The keyword is followed by some or all of the following rows of data described below. The set of records is ended by a blank record containing only a slash

1. $A_g$
   DEFAULT: $A_g = 4.9402 \times 10^{-3}$

2. $B_g$
   DEFAULT: $B_g = 5.0956 \times 10^{-5}$

3. $C_g$
   DEFAULT: $C_g = 2.9223 \times 10^{-6}$

4. $D_g$
   DEFAULT: $D_g = 2.5077$

**Example**

```
THSVC
  1.3586E-2
  3.6217E-5
  2.9223E-6
  2.5077
/
```
**THTABB**

**Use tabular water densities**

This keyword can only be used with the **THERMAL** option.

The keyword requests that tabular water densities be used instead of being calculated from analytic functions (see **THANALB**).
This keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section). THWVC allows the user to specify the coefficients A\(_w\), B\(_w\) and C\(_w\) in the expression for the water viscosity

\[
\mu_w = \frac{A_w}{1 + B_w T_F + C_w T_F^2}
\]

where the temperature T\(_F\) is in °F. The default values are chosen to match the water viscosity over a large temperature range. However, if the simulation occurs over a relatively narrow temperature range the coefficients may be modified to improve this fit.

**Note:** Regardless of the unit system used for the simulation the water viscosity expression is always evaluated with the temperature in °F.

The keyword is followed by some or all of the following rows of data described below. The set of records is ended by a blank record containing only a slash

1. A\(_w\)
   
   DEFAULT: A\(_w\) = 2.1850
2. B\(_w\)
   
   DEFAULT: B\(_w\) = 0.04012
3. C\(_w\)
   
   DEFAULT: C\(_w\) = 5.1547 x 10\(^{-6}\)

**Example**

```
THWVC
  2.06E2
  3.05
  1.30E-3
/
```
TIGHTEN

Tightens simulator control parameters

The TIGHTEN keyword should be used with great care to tighten certain convergence parameters.

The data consists of one item. The keyword must be terminated by a slash (/).

1. The factor for tightening linear, non-linear and material balance convergence targets and maximums. If the value is less than 1.0 then the minimum timestep and minimum choppable timestep will also be reduced by this factor. Values greater than 1 have the effect of loosening the convergence criteria.

   DEFAULT: 1.0

Note: These values are applied only to the current convergence criteria. Any subsequent use of the TUNING keyword will reset the convergence criteria.

Example

```plaintext
TIGHTEN
0.01 /
```
TIGHTENP  Tightens simulator control parameters

The TIGHTENP keyword should be used with great care to tighten certain convergence parameters.

The data consists of four items. The keyword must be terminated by a slash (/).

1. The factor to tighten linear convergence targets and maximums by. Values greater than 1 have the effect of loosening the convergence criteria.
   DEFAULT: 1.0

2. Maximum number of linear iterations allowed
   DEFAULT: No change

3. The factor to tighten non-linear convergence targets and maximums by. Values greater than 1 have the effect of loosening the convergence criteria.
   DEFAULT: 1.0

4. Maximum number of non-linear iterations allowed.
   DEFAULT: No change

Note: These values are applied only to the current convergence criteria. Any subsequent use of the TUNING keyword will reset the convergence criteria.

Example

```
TIGHTENP 0.001 1* 0.1 1* /
```
**TIME**

Advances simulator to new report time(s)

The keyword should be followed by up to 1000 real numbers, each of which specifies the time of a report step. The data should be terminated with a slash (/).

The keyword specifies a set of time values for report steps. The time values must be increasing.

After each timestep a report of the current state of the reservoir is produced (see the **RPTSCHED** and **RPTPRINT** keywords).

UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

---

**Note:** The reports steps in the **SCHEDULE** section must be specified in chronologically increasing order. Care should be taken to ensure that this is the case especially when using different types of report time specification keywords (such as **DATES**, **TIME** and **TSTEP**) in the same simulation.

---

**Example**

```
TIME
365 730 1095 1460 1825 /
```
Specify run title

This enables a run title to be set up, which is included in the Print file headers. The syntax is rather different from that of other keywords, the line after the TITLE keyword being read directly.

No quotes or slash characters are required.

In ECLIPSE 300, the TITLE keyword may be specified more than once in the dataset, and may be changed. Any changed title will be used in all subsequent Print file headers. In addition to the use of the TITLE keyword, a primary title is read by the program when the job is submitted, and this fixed title is output in addition to any specified using the TITLE keyword. The primary title is normally supplied to the job submission macro.

Example

```
TITLE
Fifth run in series - increased production rate
```
TLMIXPAR  Todd-Longstaff mixing parameter

This keyword should only be used in runs which use the miscible flood option (keyword MISCIBLE in the RUNSPEC section) or the solvent model (keyword SOLVENT). The keyword is obligatory in these runs. It has no effect on the polymer model (keyword POLYMER) where keyword PLMIXPAR must be used instead.

The data consists of NTMISC (see keyword MISCIBLE) records of Todd-Longstaff mixing data. Each record consists of the following items of data, terminated by a slash (/).

1. Todd-Longstaff mixing parameter for the viscosity calculation.
   
   The value of the mixing parameter for each miscibility region should be in the range 0.0 to 1.0 inclusive.
   
   The mixing parameter has no associated units.

2. Todd-Longstaff mixing parameter for the density calculation.
   
   DEFAULT: Same value as the viscosity mixing parameter set in item 1.
   
   This second data item is ignored when TLMIXPAR is used in a Polymer Flood Model.

**Note:** The format of this keyword was changed for the 95A release. In pre-95A datasets, the second item was previously a character string (DEN or NODEN) instead of a real number denoting the density mixing parameter. Pre-95A datasets having the DEN / NODEN flag set must have their TLMIXPAR keyword data modified, or they will report an error. DEN corresponds to defaulting the density mixing parameter, and NODEN corresponds to setting the second mixing parameter to 0.0. No change need be made if the DEN / NODEN flag was defaulted.

**Note:** From the 2011.1 release onwards it is possible for simulations to use both the polymer and the solvent or miscible flood option. as noted above, mixing parameters for the Polymer Model should be specified with keyword PLMIXPAR.

**Example**

With NTMISC=5:

```plaintext
-- RUNSPEC section:
MISCIBLE
  5 /
-- PROPS section:
TLMIXPAR
  0.35 /
  0.57 /
  0.19 /
  0.0 /
  1.0 0.7 /
```

*eclEditor* 100 300
SPECIAL
RUNSPEC
GRID
EDIT
PROPS
REGIONS
SOLUTION
SUMMARY
SCHEDULE

--- Keywords ---

2262
Tracer concentration regions

The keyword specifies the region numbers to be used for initializing the concentration of a tracer in each grid block and specify link cells to tables of initial tracer concentration with respect to depth.

The actual keyword to be used is a concatenated name of up to 8 characters in length, which consists of the following segments:

**ECLIPSE 100**

1. Characters 1-4 must be the character string TNUM.
2. Character 5 must be the letter F or S.
   - For a tracer whose associated stock tank phase can only exist in the free state, the letter F (free) must be used. This case includes all water phase tracers, oil phase tracers unless VAPOIL is specified in the RUNSPEC section, and gas phase tracers unless DISGAS is specified in RUNSPEC.
   - For a tracer whose associated stock tank phase can exist in the solution state (for example gas when DISGAS is specified in the RUNSPEC section, and oil when VAPOIL is in RUNSPEC) the letter S (solution) is allowed. A tracer which is embedded in such a phase requires both the free and solution component concentrations to be initialized.
3. Characters 6-8 must be the name of the tracer which is to be initialized, as specified in keyword TRACER in the PROPS section. Surfactant concentrations can be input using the character string SUR and foam concentrations using FOA.

**ECLIPSE 300**

1. Characters 1-4 must be the character string TNUM.
2. Characters 5-8 must be the name of the tracer that is being initialized, as specified in keyword TRACER in the PROPS section.

The keyword should be followed by one integer for every grid block in the current input box, specifying the initial tracer concentration region number to which it belongs. The region number should not be less than 1 or greater than NTTRVD (see item 4 in keyword EQLDIMS). The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*0.22). Note that spaces must not be inserted on either side of the asterisk (*).

See "Tracer tracking" in the ECLIPSE Technical Description for further information.

These keywords can now be used in conjunction with ADD, ADDREG, MULTIPLY or MULTIREG, in addition to_EQUALS, EQUALREG and COPYBOX.

**Examples**

**Example 1**

Where DISGAS is specified; NDIVIX=8, NDIVIY=6 and NDIVIZ=3 are specified in RUNSPEC keyword DIMENS; the gas phase tracer is 'SGS; and NTTRVD ≥ 2 in EQLDIMS item 4:
Example 2

Where NDIVIX=2, NDIVIY=3 and NDIVIZ=3 are specified in RUNSPEC keyword DIMENS; the water phase tracer is 'WT1'; and NTTRVD ≥4 in EQLDIMS item 4:

```
TNUMFWT1
12*1 6*4 /
```

Example 3

```
TNUMTRAA
48*2
48*3
48*1 /
```

ECLIPSE 300
TOLCRIT

Specify tolerance for critical saturations

The way in which the initial relative permeability critical table saturations are determined can be modified by using the TOLCRIT keyword.

By default the critical saturation for a given relative permeability function is set to the last saturation table entry for which the relative permeability is notionally zero; in practice, the criterion used for zero is less than a threshold value which by default is $1.0 \times 10^{-6}$ for ECLIPSE 100 and $1.0 \times 10^{-20}$ for ECLIPSE 300.

The default threshold can be changed by specifying TOLCRIT. Relative permeability values which are lower than this threshold will be treated as zero for the purposes of determining the critical saturation. Consideration should be given to the choice of this threshold value when modeling saturation functions with low initial values of relative permeability and when comparing results between simulators.

The keyword is followed by a single item and terminated by a slash (/):

Tolerance used to determine critical saturations

Values should be between 0 and 1.

DEFAULT:

1.0E-6 (ECLIPSE 100)
1.0E-20 (ECLIPSE 300)

See "Saturation table scaling" in the ECLIPSE Technical Description for more information.

Example

```
TOLCRIT 1.0E-8 /
```
**TOPS**

**Depths of the top face of each grid block**

The keyword should be followed by one number for every grid block in the current input box, specifying the depth at the top of each grid block.

The data must be terminated by a slash (/).

**UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

Every TOPS value in the top plane (K = 1) must be specified in one way or another by the end of the GRID section. Values in lower planes (K > 1) that are not specified are computed by adding the gross grid block thickness (DZ) to the TOPS value for the corresponding grid block in the plane above.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0.127). Note that spaces must not be inserted on either side of the asterisk.

**Example**

```
----- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX 1 8 13 1 1 /
TOPS
6222 6232 6242 6250 6256 6260 6262 6259
6182 6192 6202 6210 6216 6221 6222 6219
6162 6172 6182 6190 6196 6201 6202 6199
6142 6152 6162 6170 6176 6181 6182 6179
6122 6132 6142 6150 6156 6161 6162 6159
6102 6112 6122 6130 6136 6141 6142 6139
6092 6102 6112 6120 6126 6130 6132 6129
6082 6092 6102 6110 6116 6120 6122 6119
6072 6082 6092 6100 6106 6110 6112 6109
6062 6072 6082 6090 6096 6100 6102 6099
6042 6052 6062 6070 6076 6080 6082 6079
6022 6032 6042 6050 6056 6060 6062 6059 /
```

**Keywords**

2266
This keyword has a number of preset tuning options. All the options set can be defined by combinations of other keywords.

The keyword contains an item of data, terminated with a slash (/). The item should be one of the following:

**DEFAULT** This is the default item; no changes are made, the current tuning setting for the simulator are used.

**AIM1** This is equivalent to the DEFAULT settings, this option is only maintained for backward compatibility reasons.

**2001A** This setting reestablishes the 2001A tuning setting, and is additionally activated by item 69 in OPTIONS3

- The limit on the timestep after a well alteration is set to 5 days for an IMPES run and 20 days for an AIM or FULLIMP run. See item 11 in the TSCRIT keyword. During history matching, quite often the report steps will be close together (for example every month), with well modifications after every report. The limit of 20 days after a well modification prevents a month timestep, and was consequently changed for the 2002A release.

- The implicit cells are controlled using the AIMCON keyword. The settings used here reproduce the 2001A settings:
  
  Set open well cells, and all cells with the same inner solver direction as implicit (item 1 in AIMCON).

  Reevaluate the set of implicit cells every 5th timestep (item 4 in AIMCON).

  Use 2001A default for implicit cell selection (item 5 in AIMCON).

  Do not select any implicit cells based on solution change (item 7 in AIMCON).

  The TOPT keyword with this item for an AIM run is equivalent to the following combination of keywords:

```
TSCRIT
-- 1-- 2-- 3-- 4-- 5-- 6-- 7-- 8-- 9--10--11--12--13-- .......
 1* 1* 1* 1* 1* 1* 1* 1* 1* 1* 1* 20 1* 1* /
AIMCON
-- 1-- 2-- 3-- 4-- 5-- 6-- 7-- 8-- 9--10--
 1 1* 1* 5 -1.0 1 -1.0 /
```

**AIMCHOP** This setting is suitable for AIM runs for which AIM1 gives high numbers of implicit cells and/or severe timestep chops. It is a similar to the AIM1 setting, but with the following modifications:

- The maximum increase factor after a timestep is 1.5 (item 4 in TSCRIT).

- The maximum decrease factor after a timestep is 0.125 (item 5 in TSCRIT).

- Cells with a normalized solution change of over a quarter (item 7 in AIMCON) of 0.1 the solution change target (item 10 in TSCRIT) is implicit, up to a maximum of 5% (item 6 in AIMCON) of the reservoir cells.
The **TOPT** keyword with this item is equivalent to the following combination of keywords:

```
SCRT
   -- 1-- 2-- 3-- 4-- 5-- 6-- 7-- 8-- 9--10--11--12--13-- .......
   1* 1* 1* 1.5 1* 1* 1* 1* .1 1* 1* 1* 1* /
AIMCON
   -- 1-- 2-- 3-- 4-- 5-- 6-- 7-- 8-- 9--10--
   2 1* 1* 1 1 1 1 1 1 1 1 1 1 /
```

**AIMLOOSE** This setting is suitable for AIM runs where large areas of the reservoir may need to be implicit (for example for a sector model with many small cells near a gas injector). It is similar to the AIM1 setting, but with the following modifications:

- No limits are placed on the number of implicit cells (items 6 and 8 of the AIMCON keyword). If large numbers of cells are made implicit this increases the memory requirements.

- The **TOPT** keyword with this item is equivalent to the following combination of keywords

```
AIMCON
   -- 1-- 2-- 3-- 4-- 5-- 6-- 7-- 8-- 9--10--
   2 1* 1* 1 1 1 1 1 1 1 1 /
```

---

**Note:** This option only changes the values specified, and does not reset all the default tuning values. Consequently, tuning keywords that appear before **TOPT** in the SCHEDULE section take effect if their values are not altered by the **TOPT** keyword.

---

**Example**

```
TOPT
AIMCHOP /
```
**TPAMEPS**

### Tables of volumetric strain values

This keyword is specific to the coal bed methane option and the ROCKPAMA Palmer-Mansoori rock compaction model.

The data comprises NTCREG (see keyword REGDIMS, item 6) tables of volumetric strain values to be used in the Palmer-Mansoori model as a function of the coal gas concentrations. The table consists of NCOMPS +1 columns of data in ECLIPSE 300 and 2 columns of data in ECLIPSE 100. If the SOLVENT model is activated in ECLIPSE 100, the keyword TPAMEPSS should be used to specify the solvent strain term.

**Column:**

1. The surface coal gas concentration
   
   Values should increase monotonically down the column.
   
   **UNITS:** sm$^3$/m$^3$ (METRIC), Mscf/ft$^3$ (FIELD), scc/cc (LAB)

2. The corresponding volumetrics strain term (eps)
   
   Values should be level or increasing down the column. This look up value replaces the term $\frac{\beta P}{1 + \beta P}$ in the normal Palmer-Mansoori formulation. See the ROCKPAMA keyword.
   
   **UNITS:** Dimensionless

3. \[NCOMPS +1\] The corresponding volumetric strain term for the remaining components.
   
   Values should be level or increasing down the column.
   
   **UNITS:** Dimensionless

The entire table may be defaulted, provided the table is not the first. Defaulted tables are replaced with a copy of the previous table. In a given table that is not defaulted, there must be the same number of entries in each column and the number of rows should not be less than 2 or greater than NSSFUN (see item 3 of keyword TABDIMS).

**Note:** The first column refers to the coal gas of the component, IC, with the corresponding strain value as given in columns IC+1, IC=1, ... NCOMPS, and is not the total coal gas concentration.

The coal region number COALNUM is used to look up the table.

See also the "Coal Bed Methane Model" in the ECLIPSE Technical Description for further information.

### Example

With (NCOMPS = 1 for ECLIPSE 300), NTCREG=1 specified in keyword REGDIMS and NSSFUN $\geq$ 14 specified in keyword TABDIMS:

<table>
<thead>
<tr>
<th>COALNUM COALNUM</th>
<th>EPS1</th>
<th>EPS2</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0000000000</td>
<td>0.0000000000</td>
<td></td>
</tr>
<tr>
<td>0.000600529</td>
<td>0.000201575</td>
<td></td>
</tr>
<tr>
<td>0.001175503</td>
<td>0.000396899</td>
<td></td>
</tr>
<tr>
<td>0.005022606</td>
<td>0.001765517</td>
<td></td>
</tr>
<tr>
<td>0.008499794</td>
<td>0.003103030</td>
<td></td>
</tr>
</tbody>
</table>

---

**Keywords**

2269
| 0.011049732 | 0.004151351 |
| 0.014539122 | 0.005688889 |
| 0.019051263 | 0.007876923 |
| 0.020090423 | 0.008416438 |
| 0.021249485 | 0.009035294 |
| 0.0225504474 | 0.009752381 |
| 0.02341045 | 0.010240000 |
| 0.026434767 | 0.012047059 |
| 0.027500578 | 0.012720497 |
TPAMEPSS  Tables of volumetric strain values for solvent

This keyword is specific to the Coal Bed Methane option and the ROCKPAMA Palmer-Mansoori rock compaction model.

The data comprises NTCREG (see keyword REGDIMS, item 6) tables of volumetric strain values to be used in the Palmer-Mansoori model as a function of the coal solvent concentrations.

Column:

1. The surface coal solvent concentration
   
   Values should increase monotonically down the column.
   
   UNITS: \( \text{sm}^3/\text{m}^3 \) (METRIC), \( \text{Mscf/ft}^3 \) (FIELD), \( \text{scc/cc} \) (LAB)

2. The corresponding volumetrics strain term (\( \epsilon \))
   
   Values should be level or increasing down the column. The look up value will replace the solvent strain term
   
   \[ \frac{\epsilon_s P}{1 + \beta_s P} \]
   
   used by the Palmer-Mansoori formulation. See the ROCKPAMA keyword.
   
   UNITS: Dimensionless

The entire table may be defaulted, provided the table is not the first. Defaulted tables are replaced with a copy of the previous table. In a given table that is not defaulted, there must be the same number of entries in each column and the number of rows should not be less than 2 or greater than NSSFUN (see item 3 of keyword TABDIMS).

The coal region number COALNUM is used to look up the table.

See also the "Coal bed methane model" in the ECLIPSE Technical Description for further information.

Example

With NTCREG=1 specified in keyword REGDIMS and NSSFUN\( \geq 14 \) specified in keyword TABDIMS:

<table>
<thead>
<tr>
<th>SC</th>
<th>EPS</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.002282519</td>
<td>0.000201575</td>
</tr>
<tr>
<td>0.004382436</td>
<td>0.000396899</td>
</tr>
<tr>
<td>0.016600137</td>
<td>0.001765517</td>
</tr>
<tr>
<td>0.02547928</td>
<td>0.00310303</td>
</tr>
<tr>
<td>0.031007804</td>
<td>0.004151351</td>
</tr>
<tr>
<td>0.037520858</td>
<td>0.005688889</td>
</tr>
<tr>
<td>0.044536954</td>
<td>0.007876923</td>
</tr>
<tr>
<td>0.05016525</td>
<td>0.01024</td>
</tr>
<tr>
<td>0.05354883</td>
<td>0.012047059</td>
</tr>
<tr>
<td>0.054654747</td>
<td>0.012720497</td>
</tr>
<tr>
<td>0.054654747</td>
<td>0.012720497</td>
</tr>
</tbody>
</table>

Keywords

2271
**TRACER**

Set up tracers

This keyword sets up a number of passive tracers, associating each of them with a particular fluid used in the study.

In a compositional run, the fluid is one of the hydrocarbon or water components. In a black oil model, fluid refers to one of the stock tank phases.

The keyword should be followed by one line for each tracer introduced. Each line must contain the data items listed below, and is terminated by a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. **Tracer name**
   - Up to 3 characters (ECLIPSE 100).
   - Up to 4 characters (ECLIPSE 300).
   
   _CAUTION_: Tracer names starting with ‘F’, ‘S’, ‘T’ or a number are not recommended since these may create naming ambiguities for postprocessors.

2. **The name of the fluid associated with the tracer:**
   - **ECLIPSE 100**
     - Stock tank phase name: OIL, WAT or GAS.
   - **ECLIPSE 300**
     - Component name, defined by CNUMS for a hydrocarbon component or WNUMS for a water component.

   If the tracer is associated with a water component and WNUMS has not been used, use the default name which is set as follows:

   - WATER if there is only one water component
   - WATnn if there is more than one water component, for example use WAT02 to associate a tracer with the second water component.

   _Note_: For a partitioned tracer this will be regarded as the free phase.

3. **Units for tracer amount (for example ‘kg’)**
   - **UNITS**: Arbitrary, but must be consistent with what is used in TBLK and TVDP.
   - **DEFAULT**: ‘sm3’ (METRIC), ‘stb’ or ‘Mscf’ (FIELD), ‘scc’ (LAB)

   In field units, oil and water phase tracers have ‘stb’ as default, whereas gas phase tracers have ‘Mscf’.

4. **The ‘solution’ phase of a partitioned tracer:**
   - **Stock tank phase name**: OIL, WAT, GAS or MULT.

   MULT indicates a multi-partitioned tracer, that is a tracer that partitions into two or more phases, and which is a generalization of the standard partitioned tracers. When MULT is specified the actual phases into which the tracer partitions must be specified together with the K(P) tables in keyword TRACERKM.
This item need only be supplied if partitioned tracers are being used (see keyword PARTTRAC).

**ECLIPSE 100**

5. The number of the K(P) table to use for this partitioned tracer within a wellbore.

This item need only be supplied if partitioned tracers are being used (see keywords PARTTRAC, TRACERKP and TRACERKM).

**ECLIPSE 100**

6. The phase which is used for adsorption calculation of a multi-partitioned tracer, that is a tracer with MULT specified in item 4. If adsorption is active (see keywords TRADS and ADSORP), this item specifies the phase concentration that will be used when calculating adsorption.

Stock tank phase name: OIL, WAT, GAS or ALL.

ALL indicates that adsorption will be based on the overall, pore-volume averaged tracer concentration.

DEFAULT: the phase specified in item 2.

This item need only be supplied if partitioned tracers are being used in conjunction with adsorption.

**Note:** Some reported tracer quantities can be prone to rounding errors if the model is made of cells with exceeding small pore volumes. This might happen if modeling a core when using FIELD or METRIC units. In this situation the solution is to convert to LAB units.

**Examples**

**Example 1**

**ECLIPSE 300**

```
TRACER
T1 PC1 /
T2 C4 /
/ 
```

**Example 2**

**ECLIPSE 100**

```
TRACER
DGS GAS /
IGS GAS /
SLT WAT /
SPR OIL GMS /
/ 
```

**Example 3**

**ECLIPSE 100**

Multi-partitioned tracer with adsorption based on overall concentration.

```
TRACER
-- NAME FREE PHASE UNITS SOL. PHASE K(P) NO. ADS. PHASE
WTR WAT 'KG' MULT 1 ALL /
/ 
```
TRACERKM Specifies the K(P) tables for the generalized, multi-partitioned tracer option

This keyword should only be used when either the environmental tracer option or the surfactant option is active and the PARTTRAC keyword has been entered in the RUNSPEC section. The keyword can only be used in conjunction with multi-partitioned tracers, that is, tracers with MULT specified in the 4th item of the TRACER keyword.

The data comprises NKPTMX+2 records (see item 2 of keyword PARTTRAC) of data each terminated by a slash (/).

First record

The first record consists of two items of data:

1. Three-character name of partitioned tracer as defined in the TRACER keyword.
2. Type of partition function specification. This item must currently be set to STANDARD indicating that tables of partition coefficients are entered as a function of pressure.

Second record

The second record contains a list of the phases that the tracer partitions into. The options are:

O (oleic), A (aqueous) and G (gaseous).

The first phase specified in the list is taken as the reference phase and must correspond to the phase specified in the item 2 of the TRACER keyword.

Remaining records

The following NKPTMX records hold tables of partition coefficients for each region. Each table, having at most NPKPMX (item 3 of keyword PARTTRAC) rows of data, applies to its corresponding partitioning region, defined by keyword TRKPF in the REGIONS section.

Each table consists of two or more columns, and is terminated by a slash (/). The number of columns must equal the number of phases specified in the second record. Column

1. Pressure
   
   Values must be monotonically increasing.
   
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2. Value of partitioning functions K(P).

Each column defines the partitioning functions between a phase and the reference phase in the order that the phases are defined in the second record. For example, if a tracer partitions between three phases there will be two columns of partitioning functions.

Example

With NKPTMX=3 for partitioned tracer PT1 partitioning into three phases:

```
TRACERKM
PT1 STANDARD /
```
<table>
<thead>
<tr>
<th></th>
<th>A</th>
<th>O</th>
<th>G</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>1.1</td>
<td>5.7</td>
<td></td>
</tr>
<tr>
<td>50</td>
<td>1.5</td>
<td>6.8</td>
<td></td>
</tr>
<tr>
<td>90</td>
<td>1.7</td>
<td>9.1</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>A</th>
<th>O</th>
<th>G</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>1.2</td>
<td>5.0</td>
<td></td>
</tr>
<tr>
<td>50</td>
<td>1.7</td>
<td>5.9</td>
<td></td>
</tr>
<tr>
<td>90</td>
<td>1.9</td>
<td>6.4</td>
<td></td>
</tr>
</tbody>
</table>

<!-- Default third table to second table -->
TRACERKP  Specifies the K(P) tables for the partitioned tracer option

This keyword should only be used when either the Environmental Tracer option or the Surfactant options is active and the PARTTRAC keyword has been entered in the RUNSPEC section.

The data comprises NKPTMX (item 2 of keyword PARTTRAC) tables specifying the partitioning function. Each table, having at most NPKPMX (item 3 of keyword PARTTRAC) entries, applies to its corresponding partitioning region, defined by keyword TRKPF in the REGIONS section.

Each table consists of 2 columns, and is terminated by a slash (/).

Column:
1. Pressure
   Values must be monotonically increasing
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
2. Value of partitioning function K(P)

Example

With 1 table and 3 pressure values:

```
TRACERKP
2000 0.00012
2600 0.00014
3310 0.00015
/```

Keywords

ECLIPSE Reference Manual

2276
Dimensions and options for tracers

The data consists of some or all of the following items of data which describe the tracers employed during the run and options for the tracer tracking algorithm (see "Tracer tracking" in the ECLIPSE Technical Description). The data must be terminated by a slash (/).

1. The maximum number of passive oil tracers entered using keyword TRACER in the PROPS section.
   DEFAULT: 0
2. The maximum number of passive water tracers entered using keyword TRACER in the PROPS section.
   DEFAULT: 0
3. The maximum number of passive gas tracers entered using keyword TRACER in the PROPS section.
   DEFAULT: 0
4. The maximum number of environmental tracers (tracers that are adsorbed or decay) if using the Environmental Tracer option (see "Environmental tracers" in the ECLIPSE Technical Description).
   DEFAULT: 0
5. Invokes the use of a numerical diffusion control algorithm for the calculation of single phase tracer concentrations in tracer tracking runs.
   DIF or SPECIAL
   Use a numerical diffusion control algorithm. If the TRACTVD keyword is present in the PROPS section, the flux limited transport scheme is used. Otherwise, the cascade algorithm will be used. When the DIF option is used when multisegment wells are present, the tracer equations still uses the standard well model and hence the multisegment well tracer summary keywords are not relevant. The DIF option is not available in combination with PARALLEL.
   NODIF or STANDARD
   No diffusion control scheme is used, that is ECLIPSE’s standard fully implicit solver is used.
   DEFAULT: NODIF

Note: When TRACITVD is used to activate implicit TVD scheme, this parameter is ignored.

6. The maximum number of non-linear iterations to be used by the tracer solution method.
   DEFAULT: 12
7. The minimum number of non-linear iterations to be used by the tracer solution method.
   DEFAULT: 1

Note: The following items should only be used with great care and after reading "Treating passive tracers as non-linear" in the ECLIPSE Technical Description.

8. Treat passive tracer as non-linear?
   DEFAULT: ‘NO’
9. One-off linear convergence tightening factor.
DEFAULT: Will not be applied

10. One-off non-linear convergence tightening factor.
    DEFAULT: Will not be applied

11. Subsequent convergence tightening factors (applied to both linear and non-linear values, timestep
    limits will also be modified; see the TIGHTEN keyword).
    DEFAULT: 1.0

12. Number of times item 11 may be applied.
    DEFAULT: 0

**Example**

Five passive water tracers

```
TRACERS
  1*  5  1*  0 DIFF 15 /
```
Requests implicit flux limited transport for tracers

This keyword requests the use of the implicit flux limited transport scheme for the tracer calculation. Flux limited transport is a higher-order TVD (total variance diminishing) scheme that reduces numerical dispersion of the tracer. (The implicit TVD is implemented with three alternative limiters.)

The TRACITVD keyword is available for single phase tracers, that is water tracers and hydrocarbon tracers in phases that do not dissolve in the complementary phase. The TRACITVD keyword does not work in combination with parallel.

See "Control of numerical diffusion" in the ECLIPSE Technical Description for further information.

1. TVD flux limiter

   This must be one of the following:

   1 Van Leer limiter
   2 Minmod limiter
   3 Super Bee limiter

   DEFAULT: 1

   Note: The use of implicit TVD with large timestep can result in non-physical effects for tracers for a subset of problems. The default Van Leer limiter should handle most situations. The minmod limiter is relatively diffusive; it can be used to reduce possible tracer effects. SuperBee is the limiter that can produce the sharpest tracer fronts; however, it may potentially have larger effects. Effects can be reduced by various tuning methods (see note below).

2. A flag to select whether tracer solutions at both the previous and the current time should be used to compute TVD flux limiter. Tracer solution at the previous time should only be used when tracer moves a small distance relative to grid size within a timestep. The default is strongly recommended.

   YES uses tracer solutions at both the previous and the current time
   NO uses tracer solutions only at the previous time

   DEFAULT: YES

   Note: The default settings should be applied for most cases requesting implicit flux limiting scheme.

   Note: The implicit flux limiting TVD scheme is a useful option to problems using non-uniform grid or requiring tracer dispersion and diffusion. For such cases, the implicit TVD scheme can provide significant speed-up compared to the explicit TVD scheme. Implicit TVD can significantly improve solution accuracy over the standard fully implicit solver by reducing numerical diffusion in the tracer equation.

   Note: Timestep tuning is useful for reducing artificial effects, typified by the over/under shooting of tracer concentrations beyond physical ranges. A large timestep can cause the implicit tracer equation to fail to converge, due to the error caused by the flow equation not being adequately converged. In addition, it can
be helpful to tune with the \texttt{TIGHTEN} keyword which can help the flow equation to converge. When flow and tracer equations are converged, non-physical effects in general are greatly diminished or disappear.

\textbf{Note:} A viable alternative is to use the Minmod mode, which can be helpful to somewhat reduce such effects without using timestep tuning or tightening. Proper tuning will improve the quality of tracer solution by computing sharp fronts with little or no non-physical effects.

\section*{Examples}

\subsection*{Example 1}

The \texttt{TRACITVD} keyword can, with strong recommendation, be used with no associated data:

\begin{verbatim}
TRACITVD
\end{verbatim}

\subsection*{Example 2}

The \texttt{TRACITVD} keyword can also be used with a chosen implicit flux limiter:

\begin{verbatim}
TRACITVD
  2  YES/
\end{verbatim}
Enable the tracking option

The TRACK option enables a division of hydrocarbons in the original reservoir to be tracked through into recovered surface volumes.

The TRACK keyword contains the following items of data:

1. Specify whether the division of the fluids is to be done at the gas-oil contact, or at a specified depth, or solely on regions defined by the user with the TRACKREG keyword. The item should be GOC, DEPTH or REGONLY.
   
   DEFAULT: GOC

2. Specify the required depth if DEPTH is selected in item 1.
   
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   
   DEFAULT: 0.0

3. Request normalization of the recovered volumes. Specify N for normalization.
   
   DEFAULT: unnormalized.

4. Specify a list of components to be tracked, or ALL for all the components.
   
   DEFAULT: ALL

Example

```
TRACK
DEPTH 9640 N ALL /
```
The keyword specifies the region numbers to be used for initializing the concentration of a tracer in each grid block for the tracking of hydrocarbon components (when the Tracer Tracking option is in use).

The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest (from 1 to NDIVIX), followed by the Y axis index (from 1 to NDIVIY), and finally the Z axis index (from 1 to NDIVIZ). NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk (*).

See "Tracer tracking" in the ECLIPSE Technical Description for further information.

**Example**

With item 5 of the REGDIMS keyword set to at least two and with 144 blocks in the model:

```
TRACKREG
48*1
48*2
48*2 /
```
TRACTBC  Traction boundary conditions

This keyword is being replaced by new keywords for setting traction boundary conditions. Please refer to keyword GMTRABC.

This keyword allows traction boundary conditions to be specified on all boundaries. A traction at a boundary surface is a force/unit area. If not specified, then that boundary is defaulted to a rigid boundary unless a displacement condition is given with the keyword DISPBC or a principal stress boundary condition is given with keyword PSTRBC.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the traction boundary condition is applied
   This must be one of the following:
   I+  Specify the boundary condition on the outer I+ edge
   I-  Specify the boundary condition on the outer I- edge
   J+  Specify the boundary condition on the outer J+ edge
   J-  Specify the boundary condition on the outer J- edge
   K+  Specify the boundary condition on the outer K+ edge
   K-  Specify the boundary condition on the outer K- edge

   DEFAULT: There is no default for this item; it must be specified.
   The K- boundary is the top boundary of the simulation, K+ is the bottom.

2. An integer specifying the type of boundary condition on this face
   This must be one of the following:
   1  A total normal traction is specified on the boundary denoted in item 1. The magnitude of this normal traction is set in item 3.
   2  A given traction with X, Y, Z components. A single X, Y, Z traction component is set in item 3 and the component name is set in item 4. Separate lines are needed to set different components.

   DEFAULT: If a boundary of a cell or face is not specified with traction, stress or displacement boundary conditions, default is a rigid boundary as specified in DISPBC item 2.

3. A normal traction on this boundary, if the boundary condition type in item 2 is 1, or an X or Y or Z component of a given traction, if the boundary condition type in item 2 is 2. The component direction is set in item 4 below.

   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
DEFAULT: 0

4. A character string that gives the component of traction if the boundary type in item 2 is 2. This must be one of ‘X’, ‘Y’ or ‘Z’.
   DEFAULT: None. It must be specified if item 2 is 2

5. First block to be modified in the I direction
   DEFAULT: 1 if an I- face

6. Last block to be modified in the I direction
   DEFAULT: NX if an I+ face

7. First block to be modified in the J direction
   DEFAULT: 1 if a J- face

8. Last block to be modified in the J direction
   DEFAULT: NY if a J+ face

9. First block to be modified in the K direction
   DEFAULT: 1 if a K- face

10. Last block to be modified in the K direction
    DEFAULT: NZ if a K+ face

Example

```plaintext
-- set a 3000 psi total compressive traction on the i+, i-, j+ and j- boundaries
-- set a 3000 psi overburden pressure (compressive stress is -ve)
-- note that the ‘K-’ direction is positive downwards
TRACTBC
-- side type total
-- traction
  'I+'  1  -3000 /
  'I-'  1   3000 /
  'J+'  1  -3000 /
  'J-'  1   3000 /
  'K-'  1  -3000 /
/
```

Keywords

2284
TRACTBCL  Traction boundary conditions in a local grid refinement

This keyword is being replaced by new keywords for setting traction boundary conditions. Please refer to keyword GMTRABCL.

This keyword specifies traction boundary conditions when a coupled stress calculation is specified with the keyword GEOMECH in the RUNSPEC section.

Use of this keyword allows traction boundary conditions to be specified on all boundaries of a local grid refinement. A traction at a boundary surface is a force/unit area. If a particular boundary is not specified, then that boundary is defaulted to a rigid boundary unless a displacement condition is given with the keyword GMDISBCL or a principal stress boundary condition is given with keyword GMPSTBCL.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. A character string specifying the boundary to which the traction boundary condition is applied
   This must be one of the following:
   I+  Specify the boundary condition on the outer I+ edge
   I-  Specify the boundary condition on the outer I- edge
   J+  Specify the boundary condition on the outer J+ edge
   J-  Specify the boundary condition on the outer J- edge
   K+  Specify the boundary condition on the outer K+ edge
   K-  Specify the boundary condition on the outer K- edge

   DEFAULT: There is no default for this item; it must be specified.
   The K- boundary is the top boundary of the simulation, K+ is the bottom.

2. An integer specifying the type of boundary condition on this face This must be one of the following:
   1 A total normal traction is specified on the boundary denoted in item 1. The magnitude of this normal traction is set in item 4.

   2 A given traction with X, Y, Z components. A single X, Y, Z traction component is set in item 4 and the component name is set in item 5. Separate lines are needed to set different components.

   DEFAULT: If a boundary of a cell or face is not specified with traction, stress or displacement boundary conditions, default is a rigid boundary as specified in GMDISBCL item 2.

3. Name of the local grid refinement
4. A normal traction on this boundary, if the boundary condition type in item 2 is 1, or an X or Y or Z component of a given traction, if the boundary condition type in item 2 is 2. The component direction is set in item 5 below.

   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

   DEFAULT: 0

5. A character string that gives the component of traction if the boundary type in item 2 is 2. This must be one of ‘X’, ‘Y’ or ‘Z’.

   DEFAULT: None. It must be specified if item 2 is 2

6. First block to be modified in the I direction

   DEFAULT: 1 if an I- face

7. Last block to be modified in the I direction

   DEFAULT: NX if an I+ face

8. First block to be modified in the J direction

   DEFAULT: 1 if a J- face

9. Last block to be modified in the J direction

   DEFAULT: NY if a J+ face

10. First block to be modified in the J direction

    DEFAULT: 1 if a K- face

11. Last block to be modified in the J direction

    DEFAULT: NZ if a K+ face

Example

```bash
-- For the local grid refinement LGR1 set a 3000 psi total compressive traction on
the i+, i-, j+ and j- boundaries
-- set a 3000 psi overburden pressure (compressive stress is -ve)
-- note that the ‘K-’ direction is positive downwards
TRACTBCL
-- side type total
-- traction
  'I+'  1  'LGR1'  -3000 /
  'I-'  1  'LGR1'   3000 /
  'J+'  1  'LGR1'  -3000 /
  'J-'  1  'LGR1'   3000 /
  'K-'  1  'LGR1'  -3000 /
/```

Keywords

2286
Requests explicit flux limited transport for tracers

This keyword requests the use of the explicit flux limited transport scheme for the tracer calculation. Flux limited transport is a higher-order TVD (total variance diminishing) scheme that reduces numerical dispersion of the tracer (The explicit TVD is implemented with the Van Leer limiter).

The TRACTVD keyword is available for single phase tracers, that is water tracers and hydrocarbon tracers in phases that do not dissolve in the complementary phase. The TRACTVD keyword only supports analytical aquifer facility, but does not work in combination with PARALLEL.

To activate the TRACTVD options, item 5 must be set to 'DIFF' in the RUNSPEC section keyword TRACERS.

See "Control of numerical diffusion" in the ECLIPSE Technical Description for further information.

The TRACTVD keyword has no associated data.

Example

```
TRACTVD
```
**Tracer adsorption functions**

This keyword should only be used when the Environmental Tracer option is active (see the 4th data item in the TRACERS keyword). The TRADS keyword inputs data describing the adsorption of a tracer on to the rock formation. The keyword should be concatenated with the tracer name.

For example:

TRADSXXX is the keyword required to input adsorption data for tracer XXX as specified in the TRACER keyword.

The data comprises NTSFUN (see keyword TABDIMS) tables of tracer adsorption functions, each terminated by a slash (/), describing the adsorption of tracer by the rock formation. Each table applies to its corresponding saturation function region, defined by keyword SATNUM in the REGIONS section (if NTSFUN > 1).

Each table consists of 2 columns of data. Column:

1. The local tracer concentration in the solution surrounding the rock.
   - The values should increase monotonically down the column.
   - The units are those of tracer concentration. By default the concentrations are dimensionless, and can be thought of as:
     - UNITS: sm$^3$/sm$^3$ (METRIC), stb/stb for liquid phase tracers, Mscf/Mscf for gas phase tracers (FIELD), scc/scc (LAB)
     - If a unit has been specified for this tracer in keyword TRACER, the units employed here are:
       - UNITS: TR/sm$^3$ (METRIC), TR/stb for liquid phase tracers, TR/Mscf for gas phase tracers (FIELD), TR/scc (LAB)
       - where TR is the unit supplied for this tracer.
2. The corresponding saturated concentration of tracer adsorbed on to the rock formation.
   - These values should increase monotonically down the column.
   - Units are (for dimensionless tracers, which are thought of as volume fractions as stated above):
     - UNITS: sm$^3$/kg (METRIC), stb/lb for liquid phase tracers, Mscf/lb for gas phase tracers (FIELD), scc/gm (LAB)
     - or (for tracers given units in keyword TRACER):
       - UNITS: TR/kg (METRIC), TR/lb (FIELD), TR/gm (LAB)
       - where TR is the unit supplied for this tracer.

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NSSFUN (see keyword TABDIMS).

**CAUTION:** The first row in the adsorption tables should consist of a pair of zeroes.

**Note:** When adsorption is specified for a partitioned tracer, the adsorbed amount is calculated based on the tracer concentration in the solution phase. For the generalized, multi-partitioned tracers (that is a tracer with
MULT specified in the 4th item of the TRACER keyword) the concentration to be used in adsorption calculations can be changed by setting the 6th item of the TRACER keyword.

See "Environmental tracers" in the ECLIPSE Technical Description for further information.

Tracer adsorption functions can also be specified using the ADSORP keyword.

**Example**

With NTSFUN = 2 and NSSFUN ≥ 5 for tracers WT1 and WT2

```
TRADSWT1
 .0000 .0000
 .0003 .0005
 .0005 .0007
 .0008 .0008
 .0010 .0008 /
 .0000 .0000
 .0010 .0010 /
TRADSWT2
 .0000 .0000
 .0003 .0005
 .0005 .0007
 .0008 .0008
 .0010 .0008 /
 .0000 .0000
 .0010 .0010 /
```
Reservoir temperature range

This keyword specifies the minimum and maximum temperatures that are expected within the reservoir. These values may be specified again at different times during the simulation. A warning is issued if the temperature in the reservoir or in a well is outside these limits.

- The minimum expected reservoir temperature. The allowable lower limit for the minimum temperature is -100 °C.
  
  UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

  DEFAULT: 1 °C

- The maximum expected reservoir temperature. The allowable lower limit for the maximum temperature is 1 °C.

  UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

  DEFAULT: 2000 °C

Example

```
--in deg F.
TRANGE
120 480 /
```
## TRANGL

### Specify transmissibilities of global-local connections

This keyword can be used to specify global-local connections explicitly, rather than have ECLIPSE work them out. If this keyword is entered, then ECLIPSE does not calculate any additional global-local connections for the current refinement. This keyword must be specified when using local grid refinements in an unstructured grid (see the `EXTFIN`, `EXTHOST` and `EXTREPGL` keywords). The keyword should be specified between the corresponding `EXTFIN` and `ENDFIN` pair.

The keyword is followed by any number of records, each terminated with a slash (/). End the set of records with a slash.

The items in each record are:

1. **IL, JL, KL** The coordinates of the local cell.
2. **IG, JG, KG** The coordinates of the connecting global cell.
3. **TRAN** The transmissibility for the connection.

   **UNITS:** cP.rm³/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm³/day/atm (PVT-M)

   **DEFAULT:** 0.0

### Example

```plaintext
TRANGL
1 2 3   5 3 5  2.0 /
3 2 3   5 6 7  2.0 /
/```

---

**Keywords**

2291
TRANR

Transmissibilities in the radial direction

This keyword specifies the transmissibility values explicitly, replacing those calculated by the program. This keyword specifies the transmissibility values in the radial direction, for runs using radial geometry.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified overwrite the R-direction transmissibilities calculated by ECLIPSE for the +R face of each grid block. Thus, a value specified for block (I,J,K) is the transmissibility between blocks (I,J,K) and (I+1,J,K).

UNITs: cP.rm$^3$/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm$^3$/day/atm (PVT-M)

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

The TRANR keyword is entirely optional. Any transmissibilities that are not altered remain at the values calculated by ECLIPSE from the GRID data.

Example

```plaintext
-- IR1-IR2 JT1-JT2 KZ1-KZ2
  BOX 5 5 3 8 4 6 /
  TRANR 18*0 /
```
Transmissibilities in the azimuthal direction

This keyword specifies the transmissibility values explicitly, replacing those calculated by the program. This keyword specifies the transmissibility values in the azimuthal (theta) direction, for runs using radial geometry.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified overwrite the theta direction transmissibilities calculated by ECLIPSE for the +theta face of each grid block. Thus, a value specified for block (I,J,K) is the transmissibility between blocks (I,J,K) and (I,J+1,K).

UNITs: \( \text{cP.m}^3/\text{day/bars (METRIC)} \), \( \text{cP.rb/day/psi (FIELD)} \), \( \text{cP.rcc/hr/atm (LAB)} \), \( \text{cP.rm}^3/\text{day/atm (PVT-M)} \)

Grid blocks are ordered with the R axis index cycling fastest, followed by the theta and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

The TRANTHT keyword is entirely optional. Any transmissibilities that are not altered remain at the values calculated by ECLIPSE from the GRID data.

Example

```plaintext
-- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX 1 4 3 3 4 6 /
TRANTHT 12*0 /
```
X-direction transmissibility values

This keyword specifies the transmissibility values explicitly, replacing those calculated by the program. The TRANX keyword applies to transmissibilities in the X-direction.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified overwrite the X-direction transmissibilities calculated by ECLIPSE for the +X face of each grid block. Thus, a value specified for block (I, J, K) is the transmissibility between blocks (I, J, K) and (I+1, J, K).

**UNITS:** cP.rm³/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm³/day/atm (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

**Note:** TRANX does not affect non-neighbor connections due to faults.

The TRANX keyword is entirely optional. Any transmissibilities that are not altered remain at the values calculated by ECLIPSE from the GRID data.

**Example**

```
-- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX 5 5 3 8 4 6 /
TRANX 18*0 /
```
This keyword specifies the transmissibility values explicitly, replacing those calculated by the program. The TRANY keyword applies to transmissibilities in the Y-direction.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified overwrite the Y-direction transmissibilities calculated by ECLIPSE for the +Y face of each grid block. Thus, a value specified for block (I,J,K) is the transmissibility between blocks (I,J,K) and (I, J+1, K).

UNITs: cP.rm³/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcch/hr/atm (LAB), cP.rm³/day/atm (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

*Note: TRANY does not affect non-neighbor connections due to faults.*

The TRANY keyword is entirely optional. Any transmissibilities that are not altered remain at the values calculated by ECLIPSE from the GRID data.

**Example**

```
-- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX 1 4 3 3 4 6 /
TRANY 12*0 /
```
Z-direction transmissibility values

This keyword specifies the transmissibility values explicitly, replacing those calculated by the program. The TRANZ keyword applies to transmissibilities in the Z-direction.

The keyword should be followed by one non-negative real number for every grid block in the current input box. The data must be terminated by a slash (/).

The values specified overwrite the Z-direction transmissibilities calculated by ECLIPSE for the +Z face of each grid block. Thus, a value specified for block (I,J,K) is the transmissibility between blocks (I, J, K) and (I, J, K+1).

UNITs: cP.rm³/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm³/day/atm (PVT-M)

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*0). Note that spaces must not be inserted on either side of the asterisk.

Note: TRANZ does not affect non-neighbor connections.

The TRANZ keyword is entirely optional. Any transmissibilities that are not altered remain at the values calculated by ECLIPSE from the GRID data.

Example

```
--- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX
  1 11 1 19 7 7 /
TRANZ
  209*0 /
```
**Enable use of transport coefficients**

This keyword enables use of transport coefficients. These are entered using the \texttt{ALPHA}, \texttt{ALPHAD} and \texttt{ALPHAI} keywords, and are used to control the flow of individual components. These allow the user to boost or hold back the flow of given components as functions of their mole fractions, and hence model fingering effects, particularly in gas injection processes at adverse mobility ratios (see "Transport coefficients" in the \textit{ECLIPSE Technical Description}).

\texttt{TRCOEF} contains a data item, terminated by a slash (/), used to identify which component is used to look up the transport coefficients.

**Example**

Enable the use of transport coefficients. These will be entered as functions of the first component mole fraction.

```
TRCOEF
1 /
```
Tracer decay data

This keyword should only be used when the environmental tracer option is active (see item 4 in the TRACERS keyword). The TRDCY keyword inputs data describing the decay of a tracer. The keyword should be concatenated with the tracer name. For example:

TRDCYXXX is the keyword required to input decay data for tracer XXX, as specified in the TRACER keyword.

The data comprises NTPVT (see keyword TABDIMS) tables of tracer decay data, each terminated by a slash (/). Each table applies to its corresponding PVT region, defined by keyword PVTNUM (if NTPVT > 1).

Each table consists of one item of data for each phase that the tracer exists in.

1. The tracer decay half-life. The specified value should be greater than zero.
   
   UNITS: day (METRIC), day (FIELD), hr (LAB)
   
   DEFAULT: 1E20 days (METRIC), 1E20 days (FIELD), 1E20 hrs (LAB)

Note: When used with the partitioned tracer option, decay applies to the tracer in the solution phase. When used with the generalized, multi-partitioned tracer option (that is a tracer with MULT specified in item 4 of the TRACER keyword) independent decay half-lives can be entered for each phase that the tracer exists in.

Note: If decay is activated for a tracer together with adsorption, the amount of the tracer adsorped on the rock will also decay according to the half-life specified in TRDCY. For a multi-partitioned tracer with individual, phase specific decay parameters a separate decay half-life can be entered for the amount of adsorped tracer. The decay half-life for adsorped tracer is entered after the parameters for each phase.

See "Environmental tracers" in the ECLIPSE Technical Description for further information.

Example

With NTPVT=2 for single phase tracers WT1 and WT2 and partitioned tracers PT1, PT2 and PT3:

```
TRACER
   WT1  WAT /
   WT2  WAT /
   PT1  OIL 1*  WAT / -- two-phase standard partitioned tracer
   PT2  OIL 1*  MULT / -- two-phase multi-partitioned tracer
   PT3  WAT 1*  MULT / -- two-phase multi-partitioned tracer
/
   ...
TRDCYW1
   10.0 /
   10.0 /
TRDCYW2
   40.0 /
   40.0 /
TRDCYP1  -- decay parameters apply to solution phase
   30 / 30 /
TRDCYP2  -- separate decay parameters for each phase
   30 10 / -- no decay of adsorped tracer
   30 10 /
TRDCYP3  -- separate decay parameters for each phase
   30 10 5 / -- separate decay parameter for adsorped tracer
   30 10 5 /
```
**Tracer diffusion data**

The TRDIF keyword inputs the tracer diffusion coefficient. The keyword should be concatenated with the tracer name.

For example:

```
TRDIFXXX is the keyword required to input the diffusion coefficient for tracer XXX as specified in the TRACER keyword.
```

The data comprises NTPVT (see keyword TABDIMS) tables of diffusion data, each terminated by a slash (/). Each table applies to its corresponding PVT region, defined by keyword PVNUM (if NTPVT > 1).

Each table consists of one diffusion coefficient for each phase that the tracer exists in.

**UNITS:** m²/day (METRIC), ft²/day (FIELD), cm²/hr (LAB)

**Note:** When used with the partitioned tracer option, the diffusion applies to the solution phase. When used with the generalized, multi-partitioned tracer option (that is a tracer with MULT specified in item 4 of the TRACER keyword) independent diffusion coefficients can be entered for each phase that the tracer exists in.

See "Environmental tracers" in the ECLIPSE Technical Description for further information.

**Note:** Heterogeneous diffusion can be input by making use of the dispersion keywords DISPDIMS, TRDIS, DISPERSE and TRNHD.

**Example**

With NTPVT=2 for single phase tracers WT1 and WT2 and partitioned tracers PT1 and PT2:

```plaintext
TRACER
  WT1  WAT /
  WT2  WAT /
  PT1  OIL  1*  WAT /
  PT2  OIL  1*  MULT /

TRIFWT1
3E-5 /
3E-5 /
TRIFWT2
0.05 /
0.05 /
TRIFPT1 -- diffusion coefficients apply to solution phase
4E-5 /
4E-5 /
TRIFPT2 -- separate diffusion coefficients for each phase
5E-4  9E-4/
5E-4  9E-4/
```
The TRDIS keyword inputs the tracer dispersion table numbers. The keyword should be concatenated with the tracer name.

For example:

TRDISXXX is the keyword required to input the dispersion table numbers for tracer XXX as specified in the TRACER keyword.

**Note:** In order to activate the dispersion option for a brine tracer, the required keyword is TRISBRI.

The data comprises NTPVT (see keyword TABDIMS) tables of dispersion table numbers, each terminated by a slash (/). Each table applies to its corresponding PVT region, defined by keyword PVNUM (if NTPVT > 1). The dispersion tables themselves are entered with the DISPERSE keyword.

Each table consists of the following items of data:

1. The tracer dispersion table number for coefficient $D_1$.
2. The tracer dispersion table number for coefficient $D_2$.
3. The tracer dispersion table number for coefficient $D_3$.
4. The tracer dispersion table number for coefficient $D_4$.
5. The tracer dispersion table number for coefficient $D_5$.
7. The tracer dispersion table number for coefficient $D_7$.
8. The tracer dispersion table number for coefficient $D_8$.

**CAUTION:** Whilst for simplicity we use $(x,y,z)$ below, these are not actually the coordinates used. The coordinate system used for dispersion actually follows the rock layers; that is, we actually require data for, and do the calculations in, the $i$-direction, the $j$-direction and the $k$-direction. These may not be the same as $(x,y,z)$ coordinates.

The form of dispersion used, to which the table numbers refer, is given by the formula:

$$\left[ \begin{array}{c} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{array} \right] \cdot \left[ \begin{array}{c} D_1(\left| v_x \right|, c) \\ D_2(\left| v_y \right|, c) \\ D_3(\left| v_z \right|, c) \\ D_4(\left| v_x \right|, c) \\ D_5(\left| v_y \right|, c) \\ D_6(\left| v_z \right|, c) \\ D_7(\left| v_x \right|, c) \\ D_8(\left| v_y \right|, c) \\ D_9(\left| v_z \right|, c) \end{array} \right] \cdot \left[ \begin{array}{c} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{array} \right] \cdot S_p c$$

Eq. 3.206

where

$c$ is the tracer concentration,

$S_p$ is the host phase saturation, and

the $v$ terms are the velocities in the normal directions to the faces of the cell.
Note that for a brine tracer, if the DSPDEINT keyword has been used, then the $c$ terms in the lookup tables are replaced with the cell’s water density.

**Note:** The option does not work with two-phase or partitioned tracers.

See also keywords DISPDIMS, DISPERSE and DSPDEINT.

**Note:** This keyword can be used in conjunction with the diffusion (TRDIF) option.

### Example

With NTPVT=2 for tracers WT1 and WT2:

```
TRDISWT1
1 2 2 2 1 2 2 2 1/
1 2 2 2 1 2 2 2 1/

TRDISWT2
1 2 2 2 3 2 2 2 1/
1 1 2 3 4 5 3 2 1/
```
**Reference temperatures**

In a run with \(N_c\) components this keyword specifies the reference temperatures at which fluid densities are defined. The keyword can be used either in a run which uses the Zudkevitch Joffe equation, or in a Thermal run (keyword `THERMAL` in the RUNSPEC section).

In a live-oil thermal run, the oil component liquid densities are given by:

\[
\rho_c = \frac{\rho_{ref,c}}{\left[1 - c_{p,c}(P - P_{ref,c})\right] \left[1 + c_{T_1,c}(T - T_{ref,c})\right]} \\
\text{Eq. 3.207}
\]

where

- \(\rho_{ref,c}\) is the reference density, specified with keyword `DREF`.
- \(c_{p,c}\) is the oil component liquid compressibility, specified with keyword `CREF`.
- \(P_{ref,c}\) is the reference pressure specified with keyword `PREF`.
- \(T_{ref,c}\) is the reference temperature, specified with this keyword.
- \(c_{T_1,c}\) is the thermal expansion coefficient, specified with keyword `THERMEX1`.

There should be one value for each component:

- **UNITS:** K (METRIC), °R (FIELD), K (LAB), K (PVT-M)
- **DEFAULT:** the standard temperature set with keyword `STCOND`

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered in item 9 of keyword `TABDIMS`. The use of multiple EoS regions is discussed in "Equations of state regions" in the `ECLIPSE Technical Description`.

**Example**

```plaintext
TREF
527.4
527.4 /
```

**Keywords**

2302
Reference temperatures for surface EoS

This optional keyword supplies reference temperatures for use in the surface calculations. The default, if this is not supplied, is for these to follow the reservoir values supplied with TREF.

**UNITS:** K (METRIC), °R (FIELD), K (LAB), K (PVT-M)

**Example**

| TREFS | 527.4 527.4 / |

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered in item 10 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of State Regions" in the ECLIPSE Technical Description.
Reference temperatures

This keyword can only be used with the thermal option (keyword THERMAL in the RUNSPEC section).

TREFT is an alias for TREF. This keyword was introduced to facilitate export of thermal data from PVT i. It makes no difference to the simulator whether this keyword or the older TREF keyword is used.
TREFTS

Reference temperatures for surface EoS

This keyword can only be used with the thermal option (keyword THERMAL in the RUNSPEC section).

TREFTS is an alias for TREFS. This keyword was introduced to facilitate export of thermal data from PVT i. It makes no difference to the simulator whether this keyword or the older TREFS keyword is used.
TRKPF

Defines tracer partitioning regions

This keyword should only be used when the partitioned tracer option is active (see the PARTTRAC keyword). The TRKPF keyword inputs the region/table number which specifies the partitioning function to use for each grid block. The keyword should be concatenated with the tracer name. For example:

TRKPFXXX is the keyword required to input adsorption data for tracer XXX as specified in the TRACER keyword.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

In ECLIPSE 100 these keywords can now be used in conjunction with ADD, ADDREG, MULTIPLY or MULTIREG, in addition to EQUALS, EQUALREG and COPYBOX.

Example

With NKPTMX=2 specified in the PARTTRAC keyword; NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS; and no input BOX set:

```plaintext
TRKPFTRA
144*1 96*2 /
```
Enables speed-up of dispersion calculation for non-homogeneous diffusion

The TRNHD keyword enables simplifications to be made to the dispersion calculation when dispersion is only being used to enable nonhomogeneous constant coefficient diffusion. In effect it is used when all the tables in the matrix in TRDIS are set to constants (that is there is no dependence on velocity or concentration) and all the off-diagonal terms in the table are to be set to zero. The keyword should be concatenated with the tracer name. For example:

TRNHDXXX is the keyword required to inform ECLIPSE that the dispersion coefficients for tracer XXX as specified in the TRACER keyword are diagonal and independent of velocity and concentration.

This keyword has no associated data.

Example

```
TRNHDWT1  
TRNHDWT2
```
TRPLPORO  Use the triple porosity model

This keyword specifies that the run will use the specialist version of multi porosity which models vuggy carbonates as a triple porosity (matrix, fracture, vugs) system.

In ECLIPSE 300 this keyword activates the multi-porosity option to perform the simulation. In ECLIPSE 100 input and coding similar to the ECLIPSE 300 multi-porosity option is used, although there is no general multi-porosity treatment available in ECLIPSE 100. See "Triple porosity" in the ECLIPSE Technical Description.

The keyword should be followed by one data item, defining the number of matrix porosities required (NOT the total number of porosities - it excludes the fracture). This is the same usage as NMATRIX for standard multi porosity. For the purpose of this usage, vugs are defined as additional matrix porosities. The number of matrix porosities must be either 2 (if only vugs with connections to the fracture are modeled, so that the porosities are matrix and connected vugs) or 3 (if additional vugs connected only to the matrix are also modeled, so that the porosities are matrix, connected vugs and isolated vugs). The data item should be followed by a slash (/).

This keyword is an alternative to NMATRIX; the two keywords may not be used together in the same deck.

This model is not compatible with the use of block to block connections (BTOBALFA or BTOBALFV) or with the use of DPNUM to create a single porosity region in a dual porosity run.

**Note:** This keyword cannot be used in parallel runs (that is if the PARALLEL keyword is present).

### Example

```
TRPLPORO
3 /
```

---

**Keywords**

ECLIPSE Reference Manual

2308
TRROCK

Specifies the tracer-rock properties

This keyword should only be used when the environmental tracer option is active (see item 4 in the TRACERS keyword). The data comprises NTSFUN (see keyword TABDIMS) tables specifying the rock properties that are required for the tracer adsorption model. Each table of rock properties applies to its corresponding saturation function region, defined by keyword SATNUM (if NTSFUN > 1).

Each table consists of either 2 or 3 items of data, terminated by a slash (/):

1. The adsorption index to be used for this rock type.
   Possible values for the adsorption index are 1 or 2.
   If a value of 1 is selected, then the tracer adsorption isotherm is retraced whenever the local tracer concentration in the solution decreases.
   If a value of 2 is selected, then no tracer desorption may occur.

2. The mass density of this rock type at reservoir conditions.
   This quantity is used in the calculation of the tracer loss due to adsorption.
   UNITS: kg/rm^3 (METRIC), lb/rb (FIELD), gm/rcc (LAB)

   Note: If either the polymer or surfactant models are active, then the mass density entered here will be ignored. The rock mass density will be taken from the PLYROCK keyword if present, or otherwise from the SURFROCK keyword.

3. The adsorption initialization model (an integer, either 1 or 2).
   This item controls the initial distribution of tracer when adsorption is active.
   Two options are available:
   1. The input tracer concentration partitions between the fluid and rock, conserving the input mass of tracer. (The total mass of tracer equals the input concentration in the fluid.)
   2. The input tracer concentration will be honored and extra adsorbed tracer is added to the rock, to be in equilibrium with the input fluid concentration.

   DEFAULT: 1

The rock mass for a grid block is given by:

Rock mass = \( V \rho_r \frac{(1-\varphi)}{\varphi} \) \hspace{1cm} \text{Eq. 3.208}

where

\( V \) is the block pore volume (at reference pressure)
\( \rho_r \) is the rock density, item 2 above
\( \varphi \) is the block porosity

The rock mass can be output using 'ROCKMASS' in the RPTSOL keyword.
See also the TRADS and "Environmental tracers" in the ECLIPSE Technical Description for further information.

**Example**

With NTSFUN=3

```
TRROCK
  2  0.00012 /
  1  0.00014  1 /
  1  0.00015  2 /
```
TSCRT  Timestepping criteria

The keyword sets a number of values used to control the timestepping of the simulator through the schedule. The data consists of some or all of the following data items, terminated with a slash (/). items not specified take their default values.

1. Length of the initial timestep.
   - **UNITS:** day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
   - **DEFAULT:** 1.0 Fully implicit or AIM (all unit sets)
   - **DEFAULT:** 0.25 IMPES (all unit sets)

2. Minimum timestep.
   - **UNITS:** day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
   - **DEFAULT:** 0.02 (all unit sets)
   - **DEFAULT:** 0.0002 Thermal option (all unit sets)

   - **UNITS:** day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
   - **DEFAULT:** 50 (all unit sets)

4. Maximum step length increase factor.
   - **DEFAULT:** 2
   - **DEFAULT:** 1.5 Thermal option

5. Maximum step length decrease factor
   - **DEFAULT:** 0.5

6. Target TTE (time truncation error).
   - **DEFAULT:** 0.2 Fully implicit, AIM or IMPSAT
   - **DEFAULT:** 0.05 IMPES
   - **Thermal option**
     - **DEFAULT:** 1.0E+20

7. Maximum TTE.
   - **DEFAULT:** 10 or 100 (if OPTIONS3 item 14 > 0) Fully implicit or IMPSAT
   - **DEFAULT:** 0.5 IMPES
   - **DEFAULT:** 2.0 AIM
   - **Thermal option**
     - **DEFAULT:** 1.0E+20 (not used by default)

8. Target throughput ratio.
   - Note this value is the worst taken over all cells and flowing components.
   - **DEFAULT:** Not used Fully implicit
   - **DEFAULT:** 0.5 IMPES or AIM

DEFAULT: Not used Fully implicit
DEFAULT: 1.0 IMPES, AIM or IMPSAT

10. Target normalized solution change.
DEFAULT: 0.2 Fully implicit or AIM
DEFAULT: 0.05 IMPES

If item 14 in OPTIONS3 is set to a non-zero value, then the above default values will be increased by a factor of 10.

Thermal option
DEFAULT: 100

Changing this value will change the default value for item 13 of keyword CVCRIT.

11. Length of timestep after a well modification and is active due to well control modifications or a well opening/closing.
UNITs: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
DEFAULT: 40 Fully implicit, AIM or IMPSAT (all unit sets)
DEFAULT: 10 IMPES (all unit sets)

If item 69 in OPTIONS3 is set to a non-zero value, then the above default values will be halved.

Thermal option
DEFAULT: 5 for Thermal simulations (all unit sets)

12. Target pressure change.
UNITs: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
DEFAULT: 100 atm

DEFAULT: 10 or 100 (if item 14 in OPTIONS3 > 0) Fully implicit
DEFAULT: 2.0 AIM
DEFAULT: 0.5 IMPES

Thermal option
DEFAULT: 1.0E+20 (not used by default)

14. Target average throughput ratio across field (IMPSAT only)
DEFAULT: 0.25

15. Target saturation change (ECLIPSE thermal option only).
DEFAULT: 0.3

16. Maximum saturation change (ECLIPSE thermal option only).
DEFAULT: 0.9

17. Target temperature change (ECLIPSE thermal option only).
UNITs: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
DEFAULT: 1.0E+20 (not used by default)

Changing this value will change the default value for item 14 of keyword CVCRIT.
18. Maximum temperature change (ECLIPSE thermal option only).
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
   DEFAULT: 1.0E+20 (not used by default)
19. Maximum number of consecutive forced timesteps before stopping the run (ignored if <0)
   DEFAULT: -1
20. Maximum number of reduced timesteps before stopping the run (ignored if <0)
   DEFAULT: -1
21. Target solid saturation change (CO2STORE option only)
   DEFAULT: 0.05
22. Maximum solid saturation change (CO2STORE option only)
   DEFAULT: 0.2

**CAUTION:** Changing these criteria can adversely affect the stability.

The TTE and solution change values are normalized to unity - that is a 0.05 solution change represents and is equivalent to a saturation change of this order.

**Note:** Data input with the TSCRIT keyword are not stored for flexible restarts. The keyword must be re-specified in the restart run (outside of any SKIPREST keywords) in order for changes to take effect.

**Example**

Set an initial timestep of 0.1 days, and a maximum timestep of 100 days.

```
TSCRIT
0.1 1* 100 /
```
Advances simulator to new report time(s)

The keyword should be followed by up to 1000 real numbers, each of which is a time interval through which the simulator is to be advanced. The data should be terminated with a slash (/).

**UNITS:** day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

After each timestep a report of the current state of the reservoir is produced (see the RPTSCHED keyword).

Note that repeat counts (for example 6*30.5) can be used if required (with no spaces before or after the asterisk).

**Example**

```
TSTEP
31 28 31 30 31 30
31 31 30 30 31 /
```
Sets simulator control parameters

The TUNING keyword should be used with great care. In general, the only parameters which may need to be altered from their default values are TSINIT, TSMAXZ and LITMAX. For example, Coning studies usually require a small timestep initially or following a significant well rate change in order to avoid a timestep chop. Changes to other parameters, and in particular to the convergence controls in section 2, are not recommended. See also the keyword NEXTSTEP, and "Convergence reports" in the ECLIPSE Technical Description.

The data comprises 3 records, each of which must be terminated by a slash (/). If a slash is encountered before the end of a record, the remaining items in the record are not altered.

**ECLIPSE 100**

Default TUNING parameters are reset by the RUNSPEC section keywords IMPES and IMPLICIT.

**Record 1: Timestepping controls**

1. **TSINIT** Maximum length of next timestep  
   **UNITS:** day (METRIC), day (FIELD), hr (LAB)  
   **DEFAULT:** 1.0 (all unit sets)
2. **TSMAXZ** Maximum length of timesteps after the next  
   **UNITS:** day (METRIC), day (FIELD), hr (LAB)  
   **DEFAULT:** 365.0 (all unit sets)
3. **TSMINZ** Minimum length of all timesteps  
   **UNITS:** day (METRIC), day (FIELD), hr (LAB)  
   **DEFAULT:** 0.1 (all unit sets)
4. **TSMCHP** Minimum choppable timestep  
   **UNITS:** day (METRIC), day (FIELD), hr (LAB)  
   **DEFAULT:** 0.15 (all unit sets)
5. **TSFMAX** Maximum timestep increase factor  
   **DEFAULT:** 3.0
6. **TSFMIN** Minimum timestep cutback factor  
   **DEFAULT:** 0.3
7. **TSFCNV** Factor by which timestep is cut after convergence failure  
   **DEFAULT:** 0.1
8. **TFDIFF** Maximum increase factor after a convergence failure  
   **DEFAULT:** 1.25
9. **THRUPT** Maximum throughput ratio  
   **DEFAULT:** 1E20 (Implicit), 0.2 (IMPES)
10. **TMAXWC**: Maximum length of the next timestep following a well modification
    
    **UNITS**: day (METRIC), day (FIELD), hr (LAB)
    
    **DEFAULT**: No limit

**Record 2: Time truncation and convergence controls**

1. **TRGTT**: Target time truncation error
    
    **DEFAULT**: 0.1 (Implicit), 1.0 (IMPES)

2. **TRGCV**: Target non-linear convergence error
    
    **DEFAULT**: 0.001 (Implicit) 0.5 (IMPES)

3. **TRGB**: Target material balance error
    
    **DEFAULT**: 1.0E-7

4. **TRGLC**: Target linear convergence error
    
    **DEFAULT**: 0.0001 (Implicit), 0.00001 (IMPES)

5. **XXTTE**: Maximum time truncation error
    
    **DEFAULT**: 10.0

6. **XXXCNV**: Maximum non-linear convergence error
    
    **DEFAULT**: 0.01 (Implicit), 0.75 (IMPES)

7. **XXMBE**: Maximum material balance error
    
    **DEFAULT**: 1.0E-6

8. **XXLCV**: Maximum linear convergence error
    
    **DEFAULT**: 0.001 (Implicit), 0.0001 (IMPES)

9. **XXWFL**: Maximum well flow rate convergence error
    
    **DEFAULT**: 0.001

10. **TRGFIP**: Target Fluid-in-place error for LGR runs
    
    **DEFAULT**: 0.025

11. **TRGSFT**: Target surfactant change (Surfactant Model only)
    
    **DEFAULT**: No limit

12. **THIONX**: Threshold for damping in ion exchange calc. (Multi-Comp. Brine Model only)
    
    **DEFAULT**: 0.01

13. **TRWGHT**: Weighting factor for active tracer updates when called from Newton Loop.
    
    (=0 fully decoupled; =1 fully coupled)
    
    **DEFAULT**: 1

The Maximum values should always be greater than the Target values. ECLIPSE 100 iterates to meet the target values but the timestep is accepted if all the maximum tolerances have been satisfied.
Record 3: Control of Newton and linear iterations

1. **NEWTMX** Maximum number of Newton iterations in a timestep
   
   DEFAULT: 12 (Implicit), 4 (IMPES)

2. **NEWTMN** Minimum number of Newton iterations in a timestep
   
   DEFAULT: 1

3. **LITMAX** Maximum number of linear iterations in a Newton iteration
   
   DEFAULT: 25

4. **LITMIN** Minimum number of linear iterations in a Newton iteration
   
   DEFAULT: 1

5. **MXWSIT** Maximum number of iterations within well flow calculation
   
   DEFAULT: 8

6. **MXWPIT** Maximum number of iterations for BHP in THP controlled wells
   
   DEFAULT: 8

7. **DDPLIM** Maximum pressure change at last Newton iteration
   
   UNITS: bars (METRIC), psi (FIELD), atm (LAB)
   
   DEFAULT: 1.0E6 (all unit sets)

8. **DDSLIM** Maximum saturation change at last Newton iteration
   
   DEFAULT: 1.0E6 (all unit sets)

9. **TRGDPR** Target maximum pressure change in a timestep
   
   UNITS: bars (METRIC), psi (FIELD), atm (LAB)
   
   DEFAULT: 1.0E6 (Implicit), 100.0 (IMPES) (all unit sets)

10. **XXXDPR** Maximum tolerable pressure change in a timestep
    
        UNITS: bars (METRIC), psi (FIELD), atm (LAB)

See also keyword **TUNINGDP**.

**Note:** The TUNING data is kept on the restart file so that it will apply to any subsequent restart run unless it is reset in the restart run. Whenever TUNING is declared with the TSINIT field defaulted, then TSINIT is set equal to 1.0.

Examples in ECLIPSE 100

**Example 1**

To set all the default values explicitly:

```
TUNING 1 365 0.1 0.15 3 0.3 0.1 1.25 0.75 /
       0.1 0.001 1E-7 0.0001
```
Example 2
To limit the next timestep to one tenth of a day, and to impose an upper limit of 10 days for all subsequent timesteps:

```
TUNING
0.1 10.0 /
/
/
```

Example 3
To set **IMPES** defaults with relaxed material balance:

```
IMPES
/
TUNING
2* 1E-5 3* 1E-4 /
/
```

Example 4
To ensure that pressure and saturation changes are small at the last Newton iteration. In this case it is also advisable to set a minimum of 3 linear iterations or to tighten the convergence of the linear equations to avoid the situation where the non-linear equations are required to be solved to a higher accuracy than the linear equations:

```
TUNING
/
/
/
LITMIN DOPLIM DDSLIM
12 1 25 3 8 8 0.0001 0.0001 /
```

Example 5
To run the Chappelear Nolen Coning study with relaxed controls:

```
TUNING
/ TIM.TRC NLN.BNC MAT.BAL LIN.CNV
0.1 0.5 0.0001 .001
10.0 1.0 0.001 .01
/
12 1 25 1 8 8 /
```

Example 6
To obtain consistency of an **IMPES** run with older versions of ECLIPSE 100 by increasing the throughput constraint to a large value, making it ineffectual:

```
TUNING
1 365 0.1 0.15 3 0.3 0.1 1.25 1.0E10 /
/
/
```
ECLIPSE 300

This keyword should be used with great care. In general, the only parameters that may need to be altered from their default values are the first three, which control the initial timestep and the minimum and maximum timestep.

This keyword duplicates some of the functionality of the TSCRIT and the CVCRIT keywords.

The TUNING keyword is copied from ECLIPSE 100 and therefore has a number of parameters that are not used in ECLIPSE 300.

These unused parameters are read in as normal and then ignored by ECLIPSE 300. The default values may differ from those in ECLIPSE 100.

The data comprises 3 sections each of which must be terminated by a slash (/). If a slash is encountered before the end of a section, the remaining items in the section are not altered.

Record 1:Timestepping controls

1. Maximum length of next timestep.
   
   UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
   
   DEFAULT: 1.0 Fully Implicit (first step), 0.5 IMPES (first step), No Action Subsequent timesteps (all unit sets)

2. Maximum length of all timesteps
   
   UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
   
   DEFAULT: 50 (all unit sets)

3. Minimum length of all timesteps
   
   UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
   
   DEFAULT: 0.02, 0.0002 (Thermal option) (all unit sets)

4. Not used.

5. Maximum timestep increase factor
   
   DEFAULT: 2.0

6. Minimum timestep cutback factor
   
   DEFAULT: 0.3

7. Not used.

8. Not used.

9. Maximum throughput ratio
   
   DEFAULT: Not used Fully Implicit
   
   DEFAULT: 1.0 IMPES, AIM or IMPSAT

10. Length of the next timestep following a well modification (same at item 11 of TSCRIT) and is active due to well control modifications or a well opening/closing.
   
   UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
DEFAULT: 40 Fully implicit, AIM or IMPSAT; 10 IMPES; 5 for Thermal simulations

If item 69 in OPTIONS3 is set to a non-zero value, then the above default values for FULLIMP, AIM, IMPSAT and IMPES will be halved.

**Record 2: Time truncation and convergence controls**

1. Target TTE (time truncation error).
   DEFAULT: 0.2  Fully Implicit, AIM or IMPSAT; 0.05 IMPES; 1.0E+20 for Thermal simulations
   DEFAULT: 0.001
3. Maximum material balance error.
   DEFAULT: 0.000001
4. Target reduction in the square of the relative residual norm in the linear solver.
   DEFAULT: Improvement by factor of 1.0E-10 in the square of the residual.
   **This is the required reduction in RMS\(^2\).**
5. Maximum TTE (Time Truncation Error)
   DEFAULT: 10 or 100 (if OPTIONS3 item 14 > 0)  Fully Implicit or IMPSAT; 0.5 IMPES; 2.0 AIM; 1.0E+20 for Thermal simulations
6. Not used.
7. Not used.
8. Not used.
9. Not used.
10. Not used.
11. Tolerance on the square of the absolute residual norm in the linear solver.
12. Target normalized solution change.
    DEFAULT: 0.2 Fully Implicit, 0.05 IMPES or AIM
    DEFAULT: 10 Fully Implicit, 0.5 IMPES or AIM
14. Maximum fugacity (phase equilibrium) error convergence criterion.
    DEFAULT: 0.001

**Record 3: Control of Newton and linear iterations**

1. Maximum number of non-linear iterations in a timestep.
   DEFAULT: 20 Fully Implicit, 15 Fully Implicit and Thermal, 6 IMPES, 12 AIM
2. Minimum number of non-linear iterations in a timestep.
   DEFAULT: 1, 2 (Thermal option)
3. Maximum number of linear iterations (LITMAX)
DEFAULT: 40, 20 if using the JALS solver or CPR solver

4. Minimum number of linear iterations in a Newton iteration
   DEFAULT: 1

5. Maximum number of iterations within well flow calculation
   DEFAULT: 40

6. Not used.

7. Maximum pressure change at last Newton iteration
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: no-limit

8. Maximum saturation change at last Newton iteration.
   DEFAULT: no-limit

9. Target maximum pressure change in a timestep.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)
   DEFAULT: 100 atm

10. Not used.

Example in ECLIPSE 300

```
TUNING
  1 365 0.1 1* 3 0.3 2* 0.75 /
  4* 10 /
  12 1 25 1 3* 1E6 /
```
TUNINGDP  Resets the simulator control parameters for high throughput cases

The TUNINGDP keyword, like the TUNING keyword, should be used with great care. Using the keyword followed by just a slash (/) modifies the simulator convergence parameters, since the default parameters with the keyword present are not the same as the defaults with the keyword absent.

The convergence parameters set in this keyword have been found to be suitable in a number of high throughput cases, typically dual porosity runs. The keyword has two effects: firstly the linear equations are solved to a tighter tolerance and secondly the non-linear equations are assumed to be converged when the change in solution over a Newton iteration falls below a critical value, irrespective of the non-linear convergence error. Essentially, convergence is assumed when either the residual is small or the solution change is small. The convergence criterion on the material balance error is unaffected, and must still be satisfied to achieve convergence.

The convergence criterion on the solution change (items 3 and 4) can override the criterion on the residual, and thus has the potential to allow a poorly converged solution to be accepted with a higher than normal residual.

Note that the TRGLCV and XXXLCV parameters are also set in the TUNING keyword and are reset by keywords IMPES and IMPPLICIT.

The TUNINGDP keyword should be followed by some or all of the following items of data terminated by a slash (/). If a slash is encountered before the end of the record, the remaining items assume their default values given below.

1. TRGLCV Target linear convergence error
   DEFAULT: 0.00001 (if TUNINGDP is present)
   DEFAULT: 0.0001 (if TUNINGDP is not present)

2. XXXLCV Maximum linear convergence error
   DEFAULT: 0.0001 (if TUNINGDP is present)
   DEFAULT: 0.001 (if TUNINGDP is not present)

3. TRGDDP Maximum pressure change during a Newton iteration, allowing convergence when the residual is still above its convergence criterion.
   UNITS: bars (METRIC), psi (FIELD), atm (LAB)
   DEFAULT: 1.0 (if TUNINGDP is present) (all unit sets)
   DEFAULT: 0.0 (if TUNINGDP is not present) (all unit sets)

4. TRGDDS Maximum saturation change during a Newton iteration allowing convergence when the residual is still above its convergence criterion.
   DEFAULT: 0.01 (if TUNINGDP is present)
   DEFAULT: 0.0 (if TUNINGDP is not present)

See also keyword TUNING.
Example

Use the TUNINGDP defaults for a high throughput case:
TUNINGH can be used to reset the convergence criteria for the linear equations solved for the Gradient option. If it is not specified, the criteria are the same as for the simulation (keyword TUNING).

The keyword should be followed by some or all of the following data items, ending with a slash (/).

**GRGLCV** Target linear convergence error
- DEFAULT: 0.0001 (or TRGLCV in TUNING if specified)

**GXXLCV** Maximum linear convergence error
- DEFAULT: 0.001 (or XXXLCL in TUNING if specified)

**GMSLCV** Reduction in RMS
- DEFAULT: 1.0E-20

**LGTMIN** Minimum number of linear iterations
- DEFAULT: 1 (or LITMW in TUNING if specified)

**LGTMAX** Maximum number of linear iterations
- DEFAULT: 25 (or LITMAX in TUNING if specified)

These parameters just change the default parameters of the linear solver for the gradient calculation, independently of the values used for the Newton iterations (see "Solution of the linear equations" in the ECLIPSE Technical Description).

When using Local Grid Refinement, values specified for TUNINGH will also override data in TUNINGL; otherwise, tuning parameters specified in TUNINGL are used for the local component of the linear equations.

The **GMSLCV** criterion measures a reduction in RMS, and is deemed to be satisfied, if for each phase \( p \),

\[
\frac{\| R_p \|_2^2}{\| R_{p}^o \|_2^2} < GMSLCV \quad \text{Eq. 3.209}
\]

where

\[
\| R_p \|_2^2 = \sqrt{\frac{\sum_{i=1}^{N} \left( \frac{R_{i,p}}{PV_i} \right)^2}{N} \cdot \Delta t \cdot \bar{B}_p} \quad \text{Eq. 3.210}
\]

with

- \( R_p \) = linear residual for phase \( p \)
- \( R_{p}^o \) = initial linear residual for phase \( p \)
- \( R_{i,p} \) = residual for phase \( p \) in grid cell \( i \)
- \( PV_i \) = pore volume for cell \( i \)
\[ N \] = total number of active cells

\[ \Delta T \] = timestep

\[ \bar{B}_p \] = field average formation volume factor for phase \( p \)

**Example**

```
TUNINGH
0.001 0.01 0.00001 /
```
TUNINGL

Simulator control parameters for local grids

TUNINGL can be used to set the simulator control parameters for local grids. The keyword data is the same as for TUNING.

The data entered with keyword TUNINGL applies to all local grids.

The TUNINGS keyword may also be used to set control parameters individually for specific local grids.
Simulator control parameters for individual local grids

TUNINGS can be used to set the simulator control parameters for a nominated local grid.

Tuning data for local grids can also be set using the keyword TUNINGL; this will apply to all local grids, except for any which have their tuning data set individually with TUNINGS.

The data comprises 4 records, each of which must be terminated by a slash (/). If a slash is encountered before the end of a record, the remaining items in the record are not altered.

**Record 1**

Record 1 is the name of the local grid to which the subsequent tuning data applies.

**Records 2 to 4**

Records 2 to 4 are the same as records 1 to 3 in the TUNING and TUNINGL keywords.

**Example**

To limit the next timestep to one tenth of a day, and to impose an upper limit of 10 days for all subsequent timesteps and limit the timestep to 5 days after each well modification in local grid LGRPOW:

```
TUNINGS
LGRPOW         /  
0.1  10.0  8* 5 /  
/  
/  
```

Keywords

2327
TVDP

**Initial tracer concentration versus depth tables**

The keyword specifies the depth tables to be used for initializing the concentration of a tracer in each grid block. The actual keyword to be used is a concatenated name of up to 8 characters in length, which consists of the following segments.

**Segment:**

1. Characters 1-4 must be the character string **TVDP**

2. (ECLIPSE 100 only) Character 5 must be the letter **F** or **S**.
   - For a tracer whose associated stock tank phase can only exist in the free state, the letter **F** (free) must be used. This case includes all water phase tracers, oil phase tracers unless **VAPOIL** is specified in the **RUNSPEC** section, and gas phase tracers unless **DISGAS** is specified in **RUNSPEC**.
   - For a tracer whose associated stock tank phase can exist in the solution state (for example, gas when **DISGAS** is specified in the **RUNSPEC** section, and oil when **VAPOIL** is in **RUNSPEC**) the letter **S** (solution) should be specified as well as **F**. Thus, such a tracer which can partition between two phases requires both the free and solution component concentrations to be initialized.

3. Characters 6-8 must be the name of the tracer which is to be initialized. Surfactant concentrations can be input using the character string **SUR** and foam concentrations using **FOA**.

4. Characters 5-7 must be the name of the tracer which is to be initialized

The data comprises **NTTRVD** (see keyword **EQLDIMS**) tables of initial tracer concentration versus depth, one for each tracer concentration region, to allow a particular tracer to be initialized.

Each table consists of 2 columns of data, and must be terminated by a slash (/). Column:

1. Depth values.
   - The values should increase monotonically down the column.
   - **UNITS:** m (**METRIC**), ft (**FIELD**), cm (**LAB**), m (**PVT-M**)

2. The corresponding initial tracer concentration values.
   - Values are should be \( \geq 0.0 \)

There must the same number of entries in each column of a given table. This number should not be less than 2 or greater than **NSTRVD** (see keyword **EQLDIMS**).

**ECLIPSE 100 only**

If a unit for tracer amount has been supplied in item 3 of keyword **TRACER**, then the unit of tracer concentration is

**UNITS:** TR/sm\(^3\) (**METRIC**), TR/stb for liquid phase tracers, TR/Mscf for gas phase tracers (**FIELD**), TR/scc (**LAB**)

where TR is the unit supplied for the tracer. For gas phase tracers the field units are TR/Mscf.

See also keyword **TNUM**.

**Note:** From 2011.1 onwards any non-zero concentrations specified for cells containing none of the host phase are automatically reset to zero. Previous behavior can be restored using item 165 in **OPTIONS**.
Examples

Example 1

ECLIPSE 100

Where VAPOIL specified, NTTRVD=1, NSTRVD=2, oil phase tracer VOL:

<table>
<thead>
<tr>
<th>TVDPFVOL</th>
<th>2000.0</th>
<th>0.0</th>
<th>6000.0</th>
<th>0.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>TVDPSVOL</td>
<td>2000.0</td>
<td>1.0</td>
<td>6000.0</td>
<td>1.0</td>
</tr>
</tbody>
</table>

Example 2

Where VAPOIL not specified, NTTRVD=2, NSTRVD=2, oil phase tracer OR1:

<table>
<thead>
<tr>
<th>TVDPFOR1</th>
<th>3000.0</th>
<th>1.0</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>4000.0</td>
<td>1.0</td>
</tr>
<tr>
<td></td>
<td>4000.0</td>
<td>0.0</td>
</tr>
<tr>
<td></td>
<td>5000.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

Example 3

ECLIPSE 300

Where NTTRVD=1, NSTRVD=3, Tracer TRA

<table>
<thead>
<tr>
<th>TVDPTRA</th>
<th>1000.0</th>
<th>0.34</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>4000.0</td>
<td>0.35</td>
</tr>
<tr>
<td></td>
<td>6000.0</td>
<td>0.39</td>
</tr>
</tbody>
</table>
Controls the transition zone option

The TZONE keyword can only be used when the end-point scaling option is active (see keywords ENDSCALE in the RUNSPEC section).

The keyword should be followed by up to 3 logical switches, either T for True or F for False (no quotes). The three switches apply to the three phases: oil, water and gas.

If a phase switch is set True then the critical saturations for that phase will be modified to be the initial immobile saturation in regions where the saturation is below the input critical value.

If the oil phase switch is set True, ECLIPSE modifies SOWCR only for oil-water or oil-water-miscible gas runs, and SOGCR is only modified in oil-gas runs.

If the water phase switch is set True, ECLIPSE modifies SWCR.

If the gas phase switch is set True, ECLIPSE modifies SGCR only for gas-water or oil-gas runs.

Output of the processed end points used in the simulation can be requested with SOWCR, SOGCR, SWCR and SGCR in the RPTSL and RPTRST keywords, but the output is headed by arguments PSOWCR, PSOGCR, PSWCR and PSGCR respectively.

See "Saturation table scaling" in the ECLIPSE Technical Description for details of the endpoint scaling procedure.

Example

```
TZONE
F T F /
```
This section contains an alphabetic listing of keywords starting with the letter U.
Dimensions for the user defined arguments facility

The UDADIMS keyword enables users to define argument quantities in the SCHEDULE section for well keywords:

- WALKALIN, WCONINJE, WECON, WFOAM, WSALT, WSURFACT, WAPI, WCONPROD, WELLSTRE, WPOLYMER, WSOLVENT, WTRACER, WELDRA, WTADD, WIMULT, WINJEDET, WECONCMF, WINJTEMP

and group keywords:

- GCONPRI, GCONPROD, GCONINJE, GECON, GCONSALE, GCONSUMP, GRUPFUEL, GRUPSALE, GTADD, GTMULT.

and segment keyword:

- WSEGTABLE

and connection keywords:

- CECON, CSKIN

and keyword:

- LINCOM.

The User Defined Arguments (UDAs) are specified by a corresponding UDQ, which can either be constants, or be composed of ECLIPSE summary keywords in conjunction with a number of mathematical functions. The required values for UDAs are taken from the corresponding UDQs. A UDA can only be used after the initialization of the corresponding UDQ.

The data consists of some or all of the following items which specify dimensions associated with the UDAs. The data must be terminated by a slash (/).

1. The number of keyword arguments in which UDQs replace numeric values.
   
   DEFAULT: 0

2. item 2 is retained for the sake of back-compatibility.
   
   DEFAULT: 0

3. The total number of unique instances in which a UDQ is used in a keyword argument.
   
   DEFAULT: 100

item 3 differs from item 1 in the sense that if only the oil rate argument of WCONPROD is specified by a UDQ, then item 1 equals 1. But if this argument is used separately in two lines of WCONPROD data, then item 3 equals 2.

*Note:* Only well, group or field UDQs are acceptable in the user defined arguments.

*Note:* In the pre-2010.1 version, items 1 and 2 were used for well and group UDAs respectively.

---

**Keywords**

2332
Example

RUNSPEC

UDQDIMS
12 12 2 2 2 2 0 2 /
UDQPARAM
2 1e19 -1.0 /
UDADIMS
2 1* 1 /

SCHEDULE

UDQ
-- A COUPLE OF FIELD QUANTITIES
DEFINE FUMW1 LOG(1+SUM(WOPR)) /
UNITS FUMW1 FLDS /
ASSIGN FUMW2 6000 /
UNITS FUMW1 FLDS /

WCONPROD
'P* 'OPEN' 'ORAT' FUMW1 FUMW2 3* 1000 /

Keywords

2333
User defined quantities

The UDQ keyword enables users to define summary quantities in the SCHEDULE section. The user defined quantities can either be constants, or composed of ECLIPSE summary quantities in conjunction with a number of mathematical functions. User defined quantities are identified by a "U" as the second character of the name. Connection, field, group, region, segment, well, aquifer and block quantities are permitted. With the exception of the field mnemonics these quantities are vectors with elements for each corresponding entity. The user defined summary quantities can be output by including them in the SUMMARY section using the conventional syntax. Since the user defined quantity will not be defined at this point, additional care should be taken to ensure that the quantity names match their later initializations. User defined summary quantities that are not initialized will return the value specified by item 3 of the UDQPARAM keyword.

User defined quantities can be used in two modes; either a value can be assigned to elements of the quantity, or the quantity can be defined in terms of existing ECLIPSE summary quantities. Elements of the arrays which are not assigned a value are marked as undefined. Internally these elements remain undefined, however when they are output in the form of a summary vector or tested in an ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS, ACTIONX) keyword, undefined elements are set to the value specified by item 3 of the UDQPARAM keyword.

User defined quantities can also be used for user defined arguments (UDA) in the SCHEDULE section for well keywords:

- WALKALIN, WCONINJE, WECON, WELDRAW, WFOAM, WSALT, WSURFACT, WAPI, WCONPROD, WELLSTRE, WPOLYMER, WSOLVENT, WTRACER, WTADD, WIMULT, WINJEDET, WECONCMF

and group keywords:

- GCONPRI, GCONPROD, GCONINJE, GECON, GCONSUMP, GGROUPFUEL, GRUPSALE, GTADD, GTMULT

and connection keywords:

- CECON, CSKIN

and keyword:

- LINCOM.

The user defined arguments are specified by a corresponding UDQ and keyword UDADIMS.

**Note:** UDAs require one timestep to take effect after the keyword containing the UDA is read.

**Note:** ECLIPSE 300 does not output the summary quantities at initial time (T=0), while ECLIPSE 100 does. Typically the undefined value, item 3 of UDQPARAM keyword, is output. It should be noted that it is possible to change the value at timestep zero by using ASSIGN and putting the UPDATE status ON. See below.

A user defined quantity is initialized by either entering an assignment or a definition, with the order of evaluation following the order of initialization. User defined quantities may refer to other user defined quantities providing they have already been initialized. Assignment differs from definition in that assignment is processed immediately, whilst a definition is evaluated after the following timestep. By default a definition will be recalculated at every timestep but this can be overridden by the user. Similarly, once a quantity is initialized units can also be specified.
The number of user defined quantities is specified in RUNSPEC using the **UDQDIMS** keyword and additional parameters can be set using the **UDQPARAM** keyword.

The **UDQ** keyword is followed by any number of records, each containing the following items of data and terminated by a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. **Operation to perform**
   - **ASSIGN** Assign a numeric value to elements of a quantity. The operation will initialize the quantity and set the update item to OFF.
   - **DEFINE** Define a mathematical expression with which to update the elements of the quantity. The operation will initialize the quantity and set the update item to ON.
   - **UNITS** Specify the units that will be written along with the summary data. The quantity must already be initialized in order to perform this operation. This data has no effect on the calculation of the quantity and is used for reporting only.
   - **UPDATE** Specify when next to evaluate the quantity definition. The quantity must already be initialized in order to perform this operation.

2. **Name of the quantity**
   - The name of the quantity must start 'CU',' FU','GU','RU','SU','WU', 'AU' or 'BU' depending on whether it is a vector of connection, field, group, region, segment, well, aquifer and block elements. Quantity names must not exceed 8 characters in length, with only the first 5 being significant in the case of region quantities. If this is an **ASSIGN** record then particular elements can be selected using the syntax outlined below.

3. **Data for operation.**
   - For an **ASSIGN** record this should be a numerical value.
   - For a **DEFINE** record this should be a mathematical expression involving a combination of the functions and data described below. The expression should evaluate to either a vector of the same type as being defined, or a scalar. If the expression evaluates to a scalar all elements of the vector will inherit this value.
   - For a **UNITS** operation this should be a character string of not more than 8 characters
   - For an **UPDATE** operation this should be one of **ON**, **OFF** or **NEXT** depending on whether the definition is to be evaluated at all, none, or just the next timestep.

### Syntax for specifying particular elements

An **ASSIGN** record can target a subset of the elements in a vector. Similarly, whilst a **DEFINE** record applies to a quantity as a whole, a subset of elements can be supplied as an argument to the functions introduced below. When a unique element of a vector is specified as an argument in a quantity definition, the argument is treated as a scalar. When a subset of a vector is supplied as an argument in a quantity definition the remaining elements in the vector remain undefined.

The following syntax should be followed.

**Connection quantities:**

- **COFR** – COFR for all connections for all wells. Treated as a vector.
COFR P1- COFR for all connections belonging to well ‘P1’. Treated as a vector.

COFR P1 1 2 3 COFR for the connection in block (1,2,3) belonging to well P1. Specifies a single element and will be treated as a scalar argument.

COFR ‘P*’ ‘-1’ ‘-1’ 3 - COFR for all connections belonging to wells with names beginning with ‘P’ in layer 3. Treated as a vector regardless of the number of matching connections.

Field quantities:

FOPR - Field quantities are always scalar arguments.

Group quantities:

GOPR - GOPR for all groups. Treated as a vector.

GOPR G1 - GOPR for group ‘G1’. Specifies a single element and will be treated as a scalar argument.

Note: In extended network modeling cases (see "The extended network model" in the ECLIPSE Technical Description), group UDQs will inherit their type (group or network group) from the right hand side of their definition. In the case of scalar assignments these UDQs are of standard group type unless previously defined as network group type.

Region quantities: Inter-region quantities such as ROFT are not permitted within user defined quantities.

ROIP - ROIP for all regions and all FIP sets, including the FIPNUM set. Treated as a vector.

ROIP_NUM - ROIP for all regions in the FIPNUM set. Treated as a vector.

ROIP_ONE - ROIP for all regions in the FIP set ‘ONE’. Treated as a vector.

ROIP_ONE 1 - ROIP for region 1 in the FIP set ‘ONE’. Specifies a single element and will be treated as a scalar argument. A FIP set name must be given if a region index is specified. Use NUM for the FIPNUM set.

Note: If a non-default region set is required, the name of the region set must be specified in the last three characters of the UDQ name, just as it would for the SUMMARY vector. So for a UDQ based on quantities in FIP set ‘ONE’, a suitable name would be RUXXXONE. Region quantities from more than one FIP set cannot be used in the same UDQ.

Segment quantities:

SOFR - SOFR for all segments belonging to all multisegment wells. Treated as a vector.

SOFR ‘P*’ - SOFR for all segments belonging to all multisegment wells with names beginning with ‘P’. Treated as a vector.

SOFR ‘P*’ 1 - SOFR for segment 1 belonging to all multisegment wells with names beginning with ‘P’. Treated as a vector regardless of the number of matching segments.
SOFR P1 1 - SOFR for segment 1 belonging to the multisegment well ‘P1’. Specifies a single element and will be treated as a scalar argument.

Well quantities:

WOPR - WOPR for all wells. Treated as a vector

WOPRP1 - WOPR for well ‘P1’. Specifies a single element and will be treated as a scalar argument.

WOPR ‘P*’ - WOPR for all wells with names beginning with ‘P’. Treated as a vector regardless of the number of matching wells.

Note: Lumped completion properties beginning with ‘W’ (for example WOPRL) are input in the same manner as well quantities except that the completion number should be appended directly to the end of the mnemonic (for example WOPRL5)

Aquifer quantities:

AAQR - AAQR for all aquifers. Treated as a vector

AAQR X - AAQR for aquifer number X. Specifies a single element and will be treated as a scalar argument.

Block quantities:

BOSAT - BOSAT for all active blocks. Treated as a vector

BOSAT X Y Z - BOSAT for block coordinates X Y Z. Specifies a single element and will be treated as a scalar argument.

Note: Well and group names containing template characters should be enclosed in quotes.

Note: Local SUMMARY keywords beginning with LB, LC or LW are not available for use in quantity definitions.

Functions available for quantity definitions

The following functions are available for use in quantity definitions. Without brackets, the functions are evaluated in order of decreasing precedence, with binary functions of equal precedence evaluated left to right and unary functions of equal precedence evaluated right to left. Brackets can be used to reorder the evaluation as required.

Binary functions take two arguments, one from the left and one from the right. Note, the majority of binary functions operate on the intersection of the two vectors, that is the output is only defined if both corresponding input elements are defined. A smaller number of the binary functions operate on the union of the two vectors, that is the output is defined if either of the corresponding input elements are defined. The binary functions are elemental, operating on each element in turn. As a consequence both the left and right functions need either to be of the same type, for example, a well vector, or one of the arguments needs to be
a scalar. If one argument is a vector and the other a scalar, the scalar value is applied to all elements in the vector.

Unary functions take a single argument usually supplied in a following bracket. The Unary functions are either elemental, or return a scalar value. The majority of the unary functions only operate on defined elements although a few also operate on undefined elements.

User defined quantity definitions are evaluated as real numbers constrained to lie in the range specified by item 2 of the `UDQPARAM` keyword. Values exceeding this range will be limited and ECLIPSE will issue a message notifying the user. Similarly, should an argument fall outside the domain of a function then the element will be returned undefined and the user notified.

<table>
<thead>
<tr>
<th>Function</th>
<th>Precedence and Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(</td>
<td></td>
<td>Opening bracket.</td>
</tr>
<tr>
<td>)</td>
<td></td>
<td>Closing bracket.</td>
</tr>
<tr>
<td>[</td>
<td></td>
<td>Start of table argument list.</td>
</tr>
<tr>
<td>]</td>
<td></td>
<td>End of table argument list.</td>
</tr>
<tr>
<td>,</td>
<td></td>
<td>Separator for table arguments.</td>
</tr>
<tr>
<td>-</td>
<td>6 unary elemental</td>
<td>Sign change of defined element.</td>
</tr>
<tr>
<td>ABS()</td>
<td>6 unary elemental</td>
<td>Absolute value of defined elements.</td>
</tr>
<tr>
<td>AVEA()</td>
<td>6 unary scalar</td>
<td>Arithmetic average of defined elements.</td>
</tr>
<tr>
<td>AVEG()</td>
<td>6 unary scalar</td>
<td>Geometric average of defined elements. The result will be undefined if the input vector contains zero or negative elements.</td>
</tr>
<tr>
<td>AVEH()</td>
<td>6 unary scalar</td>
<td>Harmonic average of defined elements. The result will be undefined if the input vector contains zero elements.</td>
</tr>
<tr>
<td>DEF()</td>
<td>6 unary elemental</td>
<td>1 if the element is defined, otherwise the result is undefined.</td>
</tr>
<tr>
<td>EXP()</td>
<td>6 unary elemental</td>
<td>Exponential of defined elements.</td>
</tr>
<tr>
<td>IDV()</td>
<td>6 unary elemental (all elements)</td>
<td>1 if the element is defined, zero if the element is undefined.</td>
</tr>
<tr>
<td>LN()</td>
<td>6 unary elemental</td>
<td>Natural logarithm of defined elements.</td>
</tr>
<tr>
<td>LOG()</td>
<td>6 unary elemental</td>
<td>Base 10 logarithm of defined elements.</td>
</tr>
<tr>
<td>MAX()</td>
<td>6 unary scalar</td>
<td>Maximum of defined elements.</td>
</tr>
<tr>
<td>MIN()</td>
<td>6 unary scalar</td>
<td>Minimum of defined elements.</td>
</tr>
<tr>
<td>NORM1()</td>
<td>6 unary scalar</td>
<td>1 norm of defined elements.</td>
</tr>
<tr>
<td>NORM2()</td>
<td>6 unary scalar</td>
<td>2 norm of defined elements.</td>
</tr>
<tr>
<td>NORMI()</td>
<td>6 unary scalar</td>
<td>Infinity norm of defined elements.</td>
</tr>
<tr>
<td>NINT()</td>
<td>6 unary elemental</td>
<td>Nearest integer to defined elements.</td>
</tr>
<tr>
<td>PROD()</td>
<td>6 unary scalar</td>
<td>Product of defined elements.</td>
</tr>
<tr>
<td>Function</td>
<td>Precedence and Type</td>
<td>Description</td>
</tr>
<tr>
<td>------------</td>
<td>--------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>RANDN()</td>
<td>6</td>
<td>unary elemental Normally distributed random number with zero mean and unit variance. The seed for the random number generator can be set using item 1 of the \texttt{UDQPARAM} keyword. The input argument is used purely to determine the size of the vector and the defined/undefined elements.</td>
</tr>
<tr>
<td>RANDU()</td>
<td>6</td>
<td>unary elemental Uniformly distributed random number between -1 and 1. The seed for the random number generator can be set using item 1 of the \texttt{UDQPARAM} keyword. The input argument is used purely to determine the size of the vector and the defined/undefined elements.</td>
</tr>
<tr>
<td>RRNDN()</td>
<td>6</td>
<td>unary elemental “Really random” normally distributed random number with zero mean and unit variance. This is the same as the \texttt{RANDN} function, but the seed for the random number generator is computed (using a unique number generated from the HH:MM:SS of the 24-hour clock) for each simulation. A base simulation case seed value will be used in restart simulations unless \texttt{UDQDIMS} item 11 is set to ‘Y’.</td>
</tr>
<tr>
<td>RRNDU()</td>
<td>6</td>
<td>unary elemental “Really random” uniformly distributed random number between -1 and 1. This is the same as the \texttt{RANDU} function, but the seed for the random number generator is computed (using a unique number generated from the HH:MM:SS of the 24-hour clock) for each simulation. A base simulation case seed value will be used in restart simulations unless \texttt{UDQDIMS} item 11 is set to ‘Y’.</td>
</tr>
<tr>
<td>SORTA()</td>
<td>6</td>
<td>unary elemental Position of the element in an ascending sort of defined elements.</td>
</tr>
<tr>
<td>SORTD()</td>
<td>6</td>
<td>unary elemental Position of the element in a descending sort of defined elements.</td>
</tr>
<tr>
<td>SUM()</td>
<td>6</td>
<td>unary scalar Sum of defined elements.</td>
</tr>
<tr>
<td>UNDEF()</td>
<td>6</td>
<td>unary elemental 1 if the element is undefined, otherwise the result is undefined.</td>
</tr>
<tr>
<td>TU*[ ]</td>
<td>6</td>
<td>unary elemental The value from the specified user defined table associated with the values of the elements in each of the arguments. The table must be defined in a \texttt{UDT} keyword, and the interpolation method used for each argument is specified in the table. There must be one argument for each dimension in the table, separated by commas. All arguments must either be the same type of vector quantity or scalar. The function returns a scalar if all arguments are scalar, or the same type of vector as its vector arguments. It is not possible to have further computations inside the square brackets - each table argument must be a simple one word mnemonic. However, table arguments can themselves be \texttt{UDQ}s if a more complicated expression is required.</td>
</tr>
<tr>
<td>&lt;=</td>
<td>5</td>
<td>binary intersection 1 if the left argument is less than or equal to the right argument, else 0. The tolerance to check for equality is specified by item 4 of the \texttt{UDQPARAM} keyword.</td>
</tr>
<tr>
<td>Function</td>
<td>Precedence and Type</td>
<td>Description</td>
</tr>
<tr>
<td>----------</td>
<td>---------------------</td>
<td>-------------</td>
</tr>
<tr>
<td>&gt;=</td>
<td>5 binary intersection</td>
<td>1 if the left argument is greater than or equal to the right argument, else 0. The tolerance to check for equality is specified by item 4 of the UDQPARAM keyword.</td>
</tr>
<tr>
<td>&lt;</td>
<td>5 binary intersection</td>
<td>1 if the left argument is less than the right argument, else 0.</td>
</tr>
<tr>
<td>&gt;</td>
<td>5 binary intersection</td>
<td>1 if the left argument is greater than the right argument, else 0.</td>
</tr>
<tr>
<td>==</td>
<td>5 binary intersection</td>
<td>1 if the left argument is equal to the right argument, else 0. The tolerance to check for equality is specified by item 4 of the UDQPARAM keyword.</td>
</tr>
<tr>
<td>!=</td>
<td>5 binary intersection</td>
<td>1 if the left argument is not equal to the right argument, else 0. The tolerance to check for equality is specified by item 4 of the UDQPARAM keyword.</td>
</tr>
<tr>
<td>^</td>
<td>4 binary intersection</td>
<td>Exponentiation.</td>
</tr>
<tr>
<td>*</td>
<td>3 binary intersection</td>
<td>Multiplication.</td>
</tr>
<tr>
<td>/</td>
<td>3 binary intersection</td>
<td>Division.</td>
</tr>
<tr>
<td>+</td>
<td>2 binary intersection</td>
<td>Addition.</td>
</tr>
<tr>
<td>-</td>
<td>2 binary intersection</td>
<td>Subtraction.</td>
</tr>
<tr>
<td>UADD</td>
<td>1 binary union</td>
<td>Union of the two vectors with intersecting elements added together.</td>
</tr>
<tr>
<td>UMAX</td>
<td>1 binary union</td>
<td>Union of the two vectors with the maximum of intersecting elements taken.</td>
</tr>
<tr>
<td>UMIN</td>
<td>1 binary union</td>
<td>Union of the two vectors with the minimum of intersecting elements taken.</td>
</tr>
<tr>
<td>UMUL</td>
<td>1 binary union</td>
<td>Union of the two vectors with intersecting elements multiplied together.</td>
</tr>
</tbody>
</table>

Table 3.50: Functions of UDQ

**Data available for quantity definitions**

Any of the quantities available to the ACTIONX keyword can be included in a quantity definition. The above syntax should be used. In addition the following simulator performance keywords are available and return scalar values.

- **ELAPSED** Returns the elapsed time in seconds.
- **MSUMLINS** Returns the total number of linear iterations since the start of the run.
- **MSUMNEWT** Returns the total number of Newton iterations since the start of the run.
- **NEWTON** Returns the number of Newton iterations used for each timestep.
TCPU  Returns the current CPU usage in seconds.
TIME  Returns the current simulation time.
TIMESTEP Returns the timestep length.

Initialization of quantities

A quantity is initialized using the first ASSIGN or DEFINE record referencing the quantity in the UDQ data. Upon initialization the elements of the quantity are undefined. When they are output or used with in an ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS, ACTIONX) keyword, undefined elements are replaced by the value specified in item 3 of UDQPARAM keyword.

Processing of quantity assignments

A quantity assignment is processed upon reading a UDQ ASSIGN record. If a subset of a vector is specified then the assignment applies only to these elements, leaving the remaining elements unaltered. Consider a simulation with four wells named P11, P12, P32 and P33. The following UDQ assignments:

\[
\begin{align*}
\text{ASSIGN WUMW1} & \quad \text{\textquoteleft P*1\textquoteleft} & 1.0 / \\
\text{ASSIGN WUMW1} & \quad \text{P12} \quad 2.0 / \\
\end{align*}
\]

would result in the user defined well vector:

\[
\begin{align*}
P11 & \quad P12 & \quad P32 & \quad P33 \\
\text{WUMW1} & \quad 1.0 & \quad 2.0 & \quad \text{UNDEFINED} & \quad \text{UNDEFINED} \\
\end{align*}
\]

Note: A quantity assignment sets the UPDATE status of the vector to OFF.

Evaluation of quantity definitions

A quantity definition is an expression that is used to update every element in the vector after every timestep. Additional consideration maybe required if a definition includes operations on subsets of vectors. Again, consider a simulation with four wells named P11, P12, P32 and P33, and the following UDQ definition:

\[
\begin{align*}
\text{DEFINE WUMW1} & \quad \text{WOPR \textquoteleft P*1\textquoteleft} + \text{WOPR \textquoteleft P*2\textquoteleft} / \\
\end{align*}
\]

Internally, the following vectors will be generated. Recall a well vector has an element for every well in the simulation and if only a subset of the wells are selected the remaining elements are left undefined:

\[
\begin{align*}
P11 & \quad P12 & \quad P32 & \quad P33 \\
\text{WOPR \textquoteleft P*1\textquoteleft} & \quad \text{WOPR P11} & \quad \text{WOPR P12} & \quad \text{UNDEFINED} & \quad \text{UNDEFINED} \\
+ & \quad \text{WOPR \textquoteleft P*2\textquoteleft} & \quad \text{UNDEFINED} & \quad \text{WOPR P12} & \quad \text{WOPR P32} & \quad \text{UNDEFINED} \\
= & \quad & \quad & & \\
\end{align*}
\]
Since the addition function acts on the intersection of the defined elements P12 is the only well for which WUMW1 is defined. All other wells would return the specified value for an undefined element. A similar result is obtained in the following example in which we accumulate on a user defined quantity:

\begin{verbatim}
ASSIGN WUMW1 P1 1 / 
DEFINE WUMW1 WUMW1 * WOPR /
\end{verbatim}

for which the calculation would be:

\begin{verbatim}
P11  P12  P32  P33
WUMW1 1  UNDEFINED  UNDEFINED  UNDEFINED
*
WOPR  WOPR P11  WOPR P12  WOPR P32  WOPR P33
=
WUMW1 WOPR P11  UNDEFINED  UNDEFINED  UNDEFINED
\end{verbatim}

The key point to note is that binary functions act sequentially on corresponding elements. If we wish to add together elements from different positions in a vector this must be done on an individual basis as scalars. For example:

\begin{verbatim}
DEFINE FUMW1 WOPR P11 + WWPR P12 /
\end{verbatim}

Alternatively, if the intention is to combine vectors then functions that act on the union of the defined elements should be considered. For example in a simulation with the FIPNUM set and two additional FIP sets named ‘ONE’ and ‘TWO’, each with the three regions, we could consider the definition:

\begin{verbatim}
DEFINE RUMW1 ROIP_NUM UMAX RGIP_ONE UMIN RWIP_TWO /
\end{verbatim}

which would generate the vectors:

\begin{verbatim}
\begin{tabular}{|c|c|c|c|c|c|c|c|}
\hline
 & NUM set & ONE set & TWO set \\
\hline
ROIP_NUM & X & X & X & U & U & U & U \\
UMAX & & & & & & & \\
RGIP_ONE & U & U & U & X & X & U & U \\
UMIN & & & & & & & \\
RWIP_TWO & U & U & U & U & X & X & X \\
\hline
\end{tabular}
\end{verbatim}

where X represents a value and U an undefined element. Note, in this case the choice of union functions is not significant since the defined portions of the vectors do not overlap.
Note: A quantity definition applies to every element in the vector. A quantity definition cannot be applied to selected elements only.

Note: A quantity definition sets the UPDATE status of the vector to ON.

Examples

SUMMARY section

Request of user defined quantities in the SUMMARY section

```plaintext
CUMW1
  P1 /
  / CUMW2
  P1 /
  /
  / FUMW1
  FUMW2
  GUMW1
  / GUMW2
  /
  / RUMW1
  /
  / RUMW1ONE
  /
  / RUMW1TWO
  /
  / WUMW1
  /
  / WUMW2
  /
  / AUMW1
  /
  / AUMW2
  1/
  / BUMW2
  1 2 3/
```

SCHEDULE section

Definition of the user defined quantities in the SCHEDULE section

```plaintext
UDQ
   -- A COUPLE OF CONNECTION QUANTITIES
   ASSIGN CUMW1 2.0 /
   ASSIGN CUMW1 P12 3.0 /
   ASSIGN CUMW1 P12 10 12 1 4.0 /
   UNITS CUMW1 CONS /
   DEFINE CUMW2 COFR/MAX(CGFR) /
   UNITS CUMW2 CONS /
   -- A COUPLE OF FIELD QUANTITIES
   DEFINE FUMW1 LOG(1+SUM(WOPR)) /
   UNITS FUMW1 FLDS /
   ASSIGN FUMW2 0 /
   DEFINE FUMW2 WOPR P1 / -- RECORD WOPR OF P1 (SCALAR) IN A FIELD UDQ
   UNITS FUMW2 FLDS /
```
How to define a cumulative quantity

UDQ

ASSIGN FU_CAPXQ 0.0 /
-- Set drilling cost for every well
ASSIGN WU_DRCST 1.0e6 /
-- Set the CAPEX as the drilling cost multiplied by the number of times it happens in a timestep
DEFINE WU_CAPEX WU_DRCST * WPWE0 /
-- Field CAPEX is the sum of well CAPEX
DEFINE FU_CAPEX SUM(WU_CAPEX) /
-- Cumulative CAPEX
ASSIGN FU_CAPEXT 0.0 /
DEFINE FU_CAPEXT FU_CAPEXT + FU_CAPEX /
DEFINE FU_CAPXQ FU_CAPEX + FU_CAPEX /
--
/
/

Note: Both FU_CAPXT and FU_CAPXQ are defined in (almost) exactly the same way but only FU_CAPEXT will give the desired behavior.

The only difference is that FU_CAPXQ is initialized at the beginning of the UDQ definitions, whereas FU_CAPEXT is initialized just before its definition. As stated above the UDQs are evaluated in the order in which they are first initialized. This means that at each timestep, FU_CAPXQ is evaluated before FU_CAPEX, whereas FU_CAPEXT is evaluated after it. Thus, when FU_CAPXQ is evaluated, the FU_CAPEX term being added is still undefined (that is 0) so nothing is added and the cumulative does not
grow. By contrast, FU_CAPXT is evaluated after FU_CAPEX (because it is first initialized after it) and so behaves as expected.

**How to use conditional UDQs**

A UDQ can take a certain value depending on a predefined condition. In the example below

```plaintext
UDQ
DEFINE FUBOOL WBHP I > 3573 /
DEFINE WUMULT ((1-FUBOOL)*2.5) +(FUBOOL*1.5)/
/
```

The value of WUMULT, 2.5 or 1.5 in this example, depends on the value of BHP in well I. The same syntax can be used for a multiple condition.

```plaintext
UDQ
DEFINE FUBL1 WBHP I > 3573 /
DEFINE FUBL2 WWIR I < 6200 /
DEFINE WUMULT (FUBL1*1)+((1-FUBL1)*((FUBL2*2)+((1-FUBL2)*3)))/
/
```

**How to use UDQs for time-derivative calculation**

As the UDQs are evaluated in the same order they have been defined or assigned, UDQs can be used to calculate a function time derivative. In the example, WUBHP2 is a timestep ahead of WUBHP1 when updating WUBHPDT, hence this later is the derivative of the BHP of the well P with regard to time.

```plaintext
UDQ
ASSIGN WUBHP1 0.0 /
DEFINE WUBHP2 WBHP P /
DEFINE WUBHPDT ((WUBHP2- WUBHP1) / TIMESTEP) /
DEFINE WUBHP1 WUBHP2 /
UNITS WUBHP1 PSIA /
UNITS WUBHP2 PSIA /
/
```
**Dimensions for the user defined quantity facility**

The data consists of some or all of the following items, specifying dimensions associated with the `UDQ` keyword. The data must be terminated by a slash (/).

1. The maximum number of functions, including brackets, permitted in a quantity definition. See the `UDQ` keyword for the list of available functions.
   - DEFAULT: 16
2. The maximum number of items permitted in a quantity definition.
   - DEFAULT: 16
3. The maximum number of user defined connection quantities.
   - DEFAULT: 0
4. The maximum number of user defined field quantities.
   - DEFAULT: 0
5. The maximum number of user defined group quantities.
   - DEFAULT: 0
6. The maximum number of user defined region quantities.
   - DEFAULT: 0
7. The maximum number of user defined segment quantities.
   - DEFAULT: 0
8. The maximum number of user defined well quantities.
   - DEFAULT: 0
9. The maximum number of user defined aquifer quantities.
   - DEFAULT: 0
10. The maximum number of user defined block quantities.
    - DEFAULT: 0
11. Specifies whether a new random number generator seed is computed for restart runs. This affects the `UDQ` functions `RRNDN` and `RRNU`. If this item is set to `Y`, then a new seed will be computed for restart simulations, hence the restart simulation random value sequence will differ from that of the base simulation. If this item is defaulted or set to `N`, then restart simulations will use the same seed as the base simulation, hence the restart simulation random value sequence will be the same as that of the base simulation.
    - DEFAULT: N

**Example**

```
UDQDIMS
8 8 2 2 2 2 2 2 2 Y /
```
Parameters for the user defined quantity facility

The data consists of some or all of the following items which specify control parameters associated with the UDQ keyword. The data must be terminated by a slash (/).

1. Integer seed value for the RANDN and RANDU random number functions. This item should be an integer greater than 0.
   DEFAULT: 1

2. The permitted range (+/-) of the user defined quantities. This item should be in the range 1.0 to 1.0E+20.
   DEFAULT: 1.0E+20

3. Value given to undefined elements when outputting data or using in an ACTION (ACTIONG, ACTIONR, ACTIONW, ACTIONS) keyword. This item should be in the range specified by item 2.
   DEFAULT: 0.0

4. The fractional equality tolerance used in the ==, !=, <= and >= functions.
   DEFAULT: 1.0E-4

Example

```
UDQPARAM
  100 1* -1.0 /
```
UDT

User defined tables

The UDT keyword enables users to define lookup tables in the SCHEDULE section. These can then be used to assign values to user defined quantities in a UDQ keyword.

Tables can have any number of dimensions between 1 and 4, any number of interpolation points per dimension, and any number of values in the table itself. The maximum number of tables, interpolation points, values and dimensions permitted in any one dataset is set using the UDTDIMS keyword.

The UDT keyword is followed by a varying number of records depending on the required dimension of the table, each terminated by a slash (/). The set of records must end with a blank record, containing only a slash (/), to indicate the end of the UDT keyword.

Record 1: Basic data for table

1. Table name
   A name, which must be unique, which will be used to identify the table in UDQ definitions. The name must begin with ‘TU’ and cannot exceed 8 characters in length.

2. Number of dimensions in this table (NUMDIM)
   An integer value between 1 and MUDTDS, the maximum number of UDT table dimensions, which is specified in item 4 in UDTDIMS. Note that it is not possible to set this value to be greater than 4.

Records 2 to NUMDIM+1

1. The type of interpolation to be used for this dimension.
   This is one of ‘NV’, ‘LC’, ‘LL’ or ‘ID’
   ‘NV’ dimensions will use the nearest value.
   ‘LC’ dimensions will be linearly interpolated within the table but clamped to the first or last value as appropriate when outside it.
   ‘LL’ dimensions will be linearly interpolated within the table and linearly extrapolated outside it.
   ‘ID’ dimensions will use the name or identifier associated with the element rather than the value. If there is no match found, the table lookup will return the undefined value specified by item 3 of the UDQPARAM keyword. The first match found will always be used.

2. The interpolation points for this dimension.
   - For ‘ID’ dimensions, these will be character strings, which can optionally use wildcard characters. This dimension type is only available for group or well mnemonics as data items.
   - For all other dimensions, these will be numerical values, which must be entered in ascending order.

Records NUMDIM+2 onwards

Each record contains multiple values, one for each interpolation point in the first dimension, terminated with a slash.
The first record corresponds to the first interpolation point in each dimension other than the first. The second record corresponds to the second interpolation point in the second dimension, and the first point for subsequent dimensions, and so on.

At the end of the records for the points for the second dimension, there is a record containing only a slash, to indicate the end of this dimension.

The next record will correspond to the first interpolation point in the second dimension, the second point in the third dimension, and the first point in the fourth dimension.

This pattern repeats, with a record containing only a slash every time we complete a dimension. So after the record which corresponds to the last point in the second dimension and the last point in the third dimension, there will be two records containing only slashes.

Note that we always have to terminate the second dimension with a slash, even if the dimension is 1 because it is a one-dimensional table.

**Examples**

**Example 1**

This is a generic example showing the structure of a four-dimensional table:

```plaintext
RUNSPEC
UDTDIMS
(max tables) (max rows) (max points) (max dims)/
...
SCHEDULE
UDT
(table name) (no. of dimensions m <= 4)/
(interpolation method IA1) (value 1) (value 2) ... (value n1)/
(interpolation method IA2) (value 1) (value 2) ... (value n2)/
(interpolation method IA3) (value 1) (value 2) ... (value n3)/
(interpolation method IA4) (value 1) (value 2) ... (value n4)/
-- the next 1..n2 rows contain n1 values each, and represent a table in dimensions IA1 and IA2, for IA3 = 1 and IA4 = 1.
-- {IA1 value 1} {IA1 value 2} ... {IA1 value i}
{Result111} {Result211} ...{Resulti11} / -- for IA2 Value 1
.../
{Result1j1} {Result2j1} ...{Resultij1} / -- for IA2 Value j
/ --Third dimension (IA3) value 1 table termination
-- Next 1 to j rows corresponding to the results table for in the dimensions of IA1 and IA2, for IA3 value 2
-- {IA1 value 1} {IA1 value 2} ... {IA1 value i}
{Result112} {Result212} ...{Resulti12} / -- for IA2 Value 1
.../
{Result1j2} {Result2j2} ...{Resultij2} / -- for IA2 Value j
/ --Third dimension (IA3) value 2 table termination
-- the next 1..n2 rows contain n1 values each, and represent a table in dimensions IA1 and IA2, for IA3 = k and IA4 = 1.
.../
--we’ve now entered all values for IA4 value 1
/
-- the next 1..n2 rows contain n1 values each, and represent a table
```

**Keywords**

ECLIPSE Reference Manual

2349
Example 2

This is a simple table with one dimension which will be linearly interpolated within the table but clamped to the first or last value as appropriate when outside it

```
RUNSPEC
  UTDIMS
  1 8 4 1 /
  ...
SCHEDULE
  UDT
    'TUTESTA' 1/  
    'LC' 4.0E+06 5.0E+06 6.0E+06 7.0E+06 /
    -- the next row contains four values, one for each point in the
    -- first and only dimension
    40 50 60 70 /
    -- this slash terminates the second dimension, which in this
    -- case is 1
    / -- this slash terminates the UDT keyword
```

Example 3

This is a sample table with 3 dimensions. The first dimension will be evaluated using linear interpolation and extrapolation, the second using the nearest value, and the third using the ID of the elements. The first dimension has three interpolation points, the other two dimensions have two each.

```
in dimensions IA1 and IA2, for IA3 = 1 and IA4 = 2.
...
-- the next 1..n2 rows contain n1 values each, and represent a table
in dimensions IA1 and IA2, for IA3 = 2 and IA4 = 2.
...
-- the next 1..n2 rows contain n1 values each, and represent a table
in dimensions IA1 and IA2, for IA3 = n3 and IA4 = 2.
...
--we’ve now entered all values for IA4 value 2
/
... 
-- the next 1..n2 rows contain n1 values each, and represent a table
in dimensions IA1 and IA2, for IA3 = 1 and IA4 = 1.
...
-- the next 1..n2 rows contain n1 values each, and represent a table
in dimensions IA1 and IA2, for IA3 = 2 and IA4 = 1.
...
-- the next 1..n2 rows contain n1 values each, and represent a table
in dimensions IA1 and IA2, for IA3 = k and IA4 = 1.
--we’ve now entered all values for IA4 value 1
/
...
--we’ve now entered all values for this table
/
--we’ve now entered all tables in this UDT keyword
```

Example 2

This is a simple table with one dimension which will be linearly interpolated within the table but clamped to the first or last value as appropriate when outside it

```
RUNSPEC
  UTDIMS
  1 8 4 1 /
  ...
SCHEDULE
  UDT
    'TUTESTA' 1/  
    'LC' 4.0E+06 5.0E+06 6.0E+06 7.0E+06 /
    -- the next row contains four values, one for each point in the
    -- first and only dimension
    40 50 60 70 /
    -- this slash terminates the second dimension, which in this
    -- case is 1
    / -- this slash terminates the UDT keyword
```

Example 3

This is a sample table with 3 dimensions. The first dimension will be evaluated using linear interpolation and extrapolation, the second using the nearest value, and the third using the ID of the elements. The first dimension has three interpolation points, the other two dimensions have two each.
So, if in a UDQ keyword we had a table reference like

```
UDQ
DEFINE WUTESTB TUTESTB[WOPR, WGPR, WWPR]
```

with wells W1 and W2, then the result of the table reference would be a well vector with two elements. We’d start off with the first element of the well vector, which is well W1.

\textit{WOPR} would be evaluated for W1, and then linear interpolation/extrapolation used. 
\textit{WGPR} would be evaluated for W1, and then the “nearest value” method of interpolation used. 
\textit{WWPR} is not evaluated at all, because the third dimension of the table uses the ‘ID’ method. Instead we use the well name for its first element, which is W1, and compare it with the names associated with the interpolation points. The name does not match the first point ‘W2’, but it does match the second point which is the wildcard character ‘*’.

So, for the third dimension, we will be using the second set of values: the two rows starting with 10 and ending with 24.

For the second dimension, we will look to see whether the value of \textit{WGPR} for W1 is closer to 2 or to 4. If it is closer to 2, we will use row 10 11 12. If it is closer to 4, we will use row 20 22 24.

And finally we will do linear interpolation or extrapolation based on where the value of \textit{WOPR} lies with relation to the interpolation points for the first dimension, which are 1 2 3.

So, if \textit{WGPR} for W1 is 2, and \textit{WOPR} for W1 is 1.5, the result would be 10.5 - we’d take row 10 11 12, and then interpolate half way between the first two values.

This process is then repeated for well W2. This time we will be using the first block of values: the two rows starting with 1 and ending with 6, since this block has the name ‘W2’. Assuming for simplicity that W2 and W1 have the same values for \textit{WGPR} and \textit{WOPR}, then this time the result will be 1.5.

We now have well vector \textit{WUTESTB} as our result, with values 10.5 and 1.5. Units can be assigned to it within the UDQ keyword as required.
Dimensions for the user defined tables facility

The data consists of the following items, which specify dimensions associated with the UDT keyword. The data must be terminated by a slash (/).

1. The maximum number of user defined tables permitted.
   DEFAULT: 0
2. The maximum number of rows permitted in any one UDT keyword.
   DEFAULT: 0
3. The maximum number of interpolation points permitted in any one dimension.
   DEFAULT: 0
4. The maximum number of dimensions permitted in any one table.
   DEFAULT: 0

To be able to define user defined tables in a UDT keyword, all items must be set to greater than 0.

Example

```
UDTDIMS
  2  20  3  2  /
```
**UNCODHMD  Indicates that gradient output is unencoded**

This keyword indicates that gradient, or sensitivity, information produced by the simulator for history matching purposes is to be unencoded.

**Note:** The “uncodedhmd” license feature is required in order to use this keyword.

This keyword has no associated data.

It affects the following areas of functionality:

- When used in conjunction with "Gradient option" in the ECLIPSE Technical Description, this keyword indicates that the HMD file produced by the Gradient option is to be unencoded. By default, the sensitivities contained in the HMD file are encoded and can only be interpreted by SimOpt. If unencoded sensitivities are required for use in an external program, an unencoded gradients license feature is also required.

- When used in conjunction with the "Adjoint gradient screening" in the ECLIPSE Technical Description, this keyword indicates that the sensitivities written to the restart files are unencoded. By default, these sensitivities are encoded.
Indicates that input files are unified

This indicates that input files (for example Restart files), which may be either multiple or unified, are unified. The default, if this keyword is not specified, is to use multiple (non-unified) files.

See "File handling in ECLIPSE" in the ECLIPSE Technical Description.

This keyword has no associated data.
Indicates that output files are unified

This indicates that output files (for example Restart and Summary files), which may be either multiple or unified, are to be unified. That is, instead of producing separate Restart and Summary files for each report step, these files are amalgamated into a single file of each type. The default is for multiple (non-unified) files.

See "File handling in ECLIPSE" in the ECLIPSE Technical Description.

This keyword has no associated data.
UNIFOUTS  Indicates that summary output files are unified

This indicates that summary output files that may be either multiple or unified, are to be unified. That is, instead of producing separate Summary files for each report step, these files are amalgamated into a single file of each type. The default is for multiple (non-unified) files. This keyword overwrites the behavior of the MULTOUT keyword, which acts on both summary and restart files.

This keyword has no associated data.
UNIFSAVE  Indicates that save files are unified

This indicates that ECLIPSE 300 Save files, which may be either multiple or unified, are to be unified. The default is multiple.

This keyword has no associated data.
USECUPL

**Tells ECLIPSE to read a reservoir coupling file**

The **USECUPL** keyword may be used in either the master run or a slave run if you wish it to read group flows or constraints from a reservoir coupling file (see "Reservoir coupling files" in the *ECLIPSE Technical Description*).

The file may be used for two purposes:

- In the master run, it can take the place of one or more of the slave reservoirs. Their slave runs are not activated, that is not specified in the **SLAVES** keyword. Instead the associated master groups are instructed in keyword **GRUPMAST** to take their production and injection rates from the reservoir coupling file.

- A slave run may be performed on its own, instead of being activated by the master run. It reads from this file the constraints that the master run originally applied to its slave groups over the course of the simulation, and apply them again in the same sequence. It expected to find the applied constraints of all its slave groups (identified with the **GRUPSLAV** keyword) on this file.

The file must previously have been written by the master run when it was controlling each of the slave reservoirs. Keyword **DUMPCUPL** instructs the Master Run to write a Reservoir Coupling file.

The **USECUPL** keyword is followed by a record containing two items of data, terminated with a slash (/). The data items are:

1. The root name of the reservoir coupling file to be read
   
   If the original coupled runs were performed over a series of restarts, there should be a sequence of reservoir coupling files, each having the root name of the data file of the master run that wrote it. In this case, you should enter the name of the last file in the sequence. Each file has a header that contains the root name of the data file from which it was restarted. Thus ECLIPSE is able to establish the chain of Reservoir Coupling files to be used, provided that they are all present. The chain can contain up to 20 restarts. ECLIPSE must be re-dimensioned if more are required.

2. The formatted/unformatted status of the input Reservoir Coupling file (a single character)
   
   **U** ECLIPSE looks for an unformatted file (.CPL)
   
   **F** ECLIPSE looks for a formatted file (.FCP)

   Note that the formatted/unformatted status of the input Reservoir Coupling file can be chosen independently of keyword **FMTIN** in the RUNSPEC section.

**Note:** The **USECUPL** instruction is not carried over on the Restart file. If you wish to restart the run and continue reading the reservoir coupling file, you must include the **USECUPL** keyword in the restart run’s SCHEDULE section. A suitable place for the keyword is at the beginning of the SCHEDULE section.

**Example**

```
USECUPL
  restart2  U /
```

**Keywords**
Use a flux file

This keyword activates the flux boundary conditions option and specifies the name of a Flux file. Only those cells in active flux regions (set with keyword FLUXREG) are active during the run.

The keyword is followed by two items of data enclosed in quotes, and terminated with a slash (/).

1. The rootname of the Flux file to be used.
2. Extrapolation flag

   YES  Allow extrapolation beyond the end of the DUMPFLUX run.
   NO   Issue an Error if the run continues beyond the end of DUMPFLUX run

   DEFAULT: NO

For further information see "Flux boundary conditions" in the ECLIPSE Technical Description.

See also the following keywords:

- FLUXNUM Set flux region numbers.
- FLUXREG Set the active flux regions.
- FLUXTYPE Specify the type of flux boundary conditions.
- THFLXTF Set the heat transmissibility factor for a non-thermal base run to a thermal reduced run.
- FLUXOPTS Specify that the reduced run is a thermal run. This keyword should be used in the non-thermal full field model if a thermal sector model will be used at a later stage.

There currently exists a number of limitations regarding the use of the flux boundary condition option, please see the "Current limitations" under the "Flux boundary conditions" section in the ECLIPSE Technical Description.

Importing ECLIPSE 100 flux files into ECLIPSE 300

Flux files generated by ECLIPSE 100 can be imported into ECLIPSE 300 provided that the pressure boundary condition has not been selected in FLUXTYPE. In order to transform oil and gas pseudo component rates generated by the black oil model into the compositional model, the compositions must be input using the following keywords:

- XSTC Oil composition at surface conditions.
- YSTC Gas composition at surface conditions.

XSTC and YSTC need only be specified when importing ECLIPSE 100 Flux files into an ECLIPSE 300 model, to convert oil and gas flows into component flows. ECLIPSE 300 Flux files already contain component flows. Flux files generated by ECLIPSE 300 cannot be imported into ECLIPSE 100.

Note: When doing a reduced run from a restart, the GRID file from the restart run must also be present in order for the simulation to initialize correctly.
Note: A thermal reduced run can only be carried out from an ECLIPSE 300 base run.

**Example**

Use a Flux file with root name `FLUX7`.

```
USEFLUX
  'FLUX7'  /
```
**USENOFLO**  Perform a useflux run with no flux file

This keyword enables a USEFLUX run to be performed without a Flux file. The flux region has no-flow boundary conditions, instead of reading the boundary flows or pressures from the Flux file.

The USEFLUX keyword should still be supplied, but the FLUX file name is ignored (and can be defaulted) if the USENOFLO keyword is present.

The USENOFLO option could be useful as a means of simulating just a section of the reservoir (for example, the drainage region of a well, for which no-flow boundary conditions may be a reasonable approximation) in isolation, without having to perform a DUMPFLUX run.

As well as imposing no-flow boundary conditions, there are two other consequences of applying the USENOFLO keyword:

- In addition to the boundary flows or pressures, the Flux file would normally supply the average pressure, $R_s$ and $R_v$ in the field. These are used, for example, to transform well flows between surface rate and reservoir volume rate. In a USENOFLO run, the average pressure, $R_s$ and $R_v$ in the flux region is used instead. Thus the correspondence between surface rate and reservoir volume rate differs from a standard USEFLUX run.

- The Flux file also contains information on the well flows outside the flux region, which is used for group control calculations when the group contains wells both within and outside the region. In a USENOFLO run this information is not present; thus group controls should not be applied in these circumstances.

See "Flux boundary conditions" in the *ECLIPSE Technical Description* for information on the Flux Boundary option.

**Example**

```
USENOFLO
```
V

This section contains an alphabetic listing of keywords starting with the letter V.
Run contains vaporized oil in wet gas

This indicates that the run contains vaporized oil in wet gas.

The keyword may be used only if both OIL and GAS are present.

If gas has a constant and uniform vaporized oil concentration, and if the pressure never drops below the dew point, then the model can be run more efficiently by omitting the OIL and VAPOIL keywords from the RUNSPEC section, treating the gas as a dry gas, and declaring a constant $R_v$ value with keyword RVCONST or RVCONSTT in the PROPS section. The simulation is run as a dry gas problem with no oil, but the calculations take into account the constant vaporized oil concentration.

The keyword has no associated data.

When using the BLACKOIL keyword in ECLIPSE 300 this option is automatically selected if the PVTG keyword in present.
**VAPPARS**

**Oil vaporization controls**

This keyword should be used only if the model allows both gas solution in the oil phase and oil vaporization in the gas phase (both keywords DISGAS and VAPOIL present in the RUNSPEC section).

The data field should contain two real numbers and should be terminated by a slash (/). Both numbers are dimensionless, and are intended to be used as parameters for history matching and sensitivity studies.

The first parameter determines the propensity of oil to vaporize in the presence of undersaturated gas. If this parameter is set to zero, the standard black oil model behavior is selected (all oil vaporizes as long as there is any undersaturated gas in the block). Larger values inhibit vaporization especially in blocks where a lot of oil has already vaporized. This parameter typically takes a value between 0 and 5.

The second parameter determines the propensity of remaining oil to get heavier (that is of $R_s$ to decrease) as lighter fractions vaporize. If this parameter is set to zero, the standard black oil model behavior is selected (the $R_s$ of remaining oil does not change as oil vaporizes). Larger values increase the differentiation between light and heavy fractions. This parameter should normally be set to a small value (less than 1).

If this keyword is not present in runs containing both live oil and wet gas, both parameters are defaulted to zero, giving standard black oil model behavior.

Note that in any system containing both dissolved gas and vaporized oil, the maximum rates of increase of $R_s$ and $R_v$ (set using DRSDT and DRVDT) are initially set to large values by ECLIPSE. The constraints imposed by the VAPPARS parameters will be disabled by the subsequent specification of either the DRSDT or DRVDT keywords. Similarly, the limits imposed by the DRSDT or DRVDT keywords will be disabled by the subsequent specification of the VAPPARS keyword.

The VAPPARS parameters ($vap1$ and $vap2$) act to reduce the maximum allowed values of $R_s$ and $R_v$ in a grid block by the relations

$$R_v = R_v \text{ sat} \left( \frac{S_o}{S_{o \text{ max}}} \right)^{vap1}$$  \hspace{1cm} \text{Eq. 3.211}

$$R_s = R_s \text{ sat} \left( \frac{S_o}{S_{o \text{ max}}} \right)^{vap2}$$  \hspace{1cm} \text{Eq. 3.212}

where

$S_{o \text{ max}}$ is the largest oil saturation reached in the grid block so far.

As $S_o$ decreases due to vaporization, further vaporization becomes more inhibited. Large values of $vap1$ and $vap2$ may cause convergence difficulties.

**Example**

```
VAPPARS
2  0.1 /
```
VCOMPACT  Compact volumes

In a run with $N_c$ components, this keyword associates a compact molar volume with each component. The compact volume is used in a modified version of the Lohrenz-Bray-Clark, see “The modified Lorentz-Bray-Clark method” in the ECLIPSE Technical Description.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of State Regions" in the ECLIPSE Technical Description.

Units: m$^3$/kg-M (METRIC), ft$^3$/lb-M (FIELD), cc/gm-M (LAB), m$^3$/kg-M (PVT-M)

**Example**

In a run with 2 components and one EoS region:

```
VCOMPACT
1.473 5.529 /
```
Critical volumes

In a run with \( N_c \) components, this keyword associates a critical molar volume with each component. These critical volumes are used in setting up the Zudkevitch-Joffe equation of state and in correlations. They are also used in the Lorentz-Bray-Clark viscosity correlation, unless separate values are entered for this purpose using the `VCRITVIS` or `ZCRITVIS` keywords.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered in item 9 of `TABDIMS`. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

UNITs: \( \text{m}^3/\text{kg-M (METRIC)} \), \( \text{ft}^3/\text{lb-M (FIELD)} \), \( \text{cc/gm-M (LAB)} \), \( \text{m}^3/\text{kg-M (PVT-M)} \)

Example

```
VCRIT
1.473 5.529 /
```
Critical volumes for detailed components

In a lumped simulation run with $N_d$ detailed components, this keyword associates a critical molar volume with each detailed component. These critical volumes are used in setting up the Zudkevitch-Joffe equation of state and in correlations. Lumped simulations are enabled with the LUMPDIMS keyword in the RUNSPEC section; further details can be found in "Delumping and lumping" in the ECLIPSE Technical Description.

The keyword should be followed by a record of $N_d$ values, terminated with a slash, for each reservoir equation of state region. The values represent the critical molar volume of each detailed component, in the order in which the detailed components are defined in the LUMPING keyword.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered in item 9 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

UNITs: m$^3$/kg-M (METRIC), ft$^3$/lb-M (FIELD), cc/gm-M (LAB), m$^3$/kg-M (PVT-M)

Example

```
VCRITDET 1.473 5.529 /
```
Critical volumes for surface EoS

This optional keyword supplies critical temperature values for use in the surface equation of state calculations. The default, if this is not supplied, is for these to follow the reservoir values supplied with VCRIT.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered in item 10 of TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.
**VCRITSDE**  
**Critical volumes for surface EoS for detailed components**

This optional keyword supplies critical temperature values for use in the surface equation of state calculations for detailed components in a lumped simulation run. The default, if this is not supplied, is for these to follow the reservoir values supplied with `VCRITDET`. Lumped simulations are enabled with the `LUMPDIMS` keyword in the `RUNSPEC` section; further details can be found in "Delumping and lumping" in the *ECLIPSE Technical Description*.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered in item 10 of `TABDIMS`. The use of multiple EoS regions is discussed in "Equations of state regions" in the *ECLIPSE Technical Description*. 

---

**Keywords**

- ECLIPSE 100
- ECLIPSE 300
- SPECIAL
- RUNSPEC
- GRID
- EDIT
- PROPS
- REGIONS
- SOLUTION
- SUMMARY
- SCHEDULE
VCRITVIS  Critical volumes for viscosity calculations

In a run with $N_c$ components, this keyword associates a critical molar volume with each component, to be used for viscosity calculations only.

If VCRITVIS or ZCRITVIS is not entered, then values entered with VCRIT or ZCRIT will be used.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered in item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

The form of VCRITVIS is the same as that of VCRIT.

UNITs: m$^3$/kg-M (METRIC), ft$^3$/lb-M (FIELD), cc/gm-M (LAB), m$^3$/kg-M (PVT-M)
**Specify velocity-dependent flow coefficient**

This keyword enables the treatment of non-Darcy effects in inter-block flow, and specifies the non-Darcy flow coefficient. This is commonly known as $\beta$, the non-Darcy flow coefficient, and is entered in units of $\text{atm} \cdot \text{s}^2 \cdot \text{g}^{-1}$ (the Forchheimer) in all unit sets. Dake [Ref. 30] shows a typical value of $\beta = 10^7 \text{cm}^{-1}$, which is 9.86 Forchheimer units.

See "Non-Darcy flow" in the ECLIPSE Technical Description for a more detailed discussion. The maximum deviation from Darcy flow is given in the REPORT section of the Print file, for example:

| Maximum deviation from Darcy flow    | 0.000259 |

**Note:** For ECLIPSE 100, this correction applies only to the gas phase and the maximum deviation is printed out if the mnemonic FFORG is used in RPTSOL and/or RPTSCHED.

**Note:** For ECLIPSE 100, by default, the non-Darcy flow calculation specified via the VDFLOW keyword (or the VDFLOWR keyword) is not applied to the well inflow calculation. This can be enabled by setting item 218 of the OPTIONS keyword to 2; in this case, for wells or well completions for which D-Factor calculations have been specified via the WDFAC keyword, the WDFACCOR keyword or item 12 of the COMPDAT keyword (or item 13 of the COMPDATL keyword), these will supersede those due to the VDFLOW or VDFLOWR keyword.

**Note:** For ECLIPSE 300, the non-Darcy flow calculation specified using the VDFLOW keyword (or the VDFLOWR keyword) is not applied to the well inflow calculations.

See also keyword VDFLOWR.

**Example**

```plaintext
VDFLOW
10.0 /
```
Specify velocity-dependent flow coefficient

This keyword is an alternative to VDFLOW allowing input of the non-Darcy flow coefficient on a region-by-region basis. The data comprises NTSFUN (see item 1 of keyword TABDIMS in the RUNSPEC section) records, each containing a single data item terminated with a slash (/).

1. The non-Darcy flow coefficient. This is commonly known as $\beta$, the non-Darcy flow coefficient, and is entered in units of atm.s$^2$.g$^{-1}$ (the Forchheimer) in all unit sets. Dake [Ref. 30] in chapter 8 shows a typical value of $\beta = 10^7$ cm$^{-1}$, which is 9.86 Forchheimer units.

See "Non-Darcy flow" in the ECLIPSE Technical Description for a more detailed discussion. The maximum deviation from Darcy flow is given in the REPORT section of the Print file, for example:

```
Maximum deviation from Darcy flow    0.000259
```

If flow occurs between two regions with differing non-Darcy flow coefficients, the average value is used.

**Note:** For ECLIPSE 100, this correction applies only to the gas phase and the maximum deviation is printed out if the mnemonic FFORG is used in RPTSOL and/or RPTSCHED.

**Note:** For ECLIPSE 100, by default, the non-Darcy flow calculation specified via the VDFLOWR keyword (or the VDFLOW keyword) is not applied to the well inflow calculation. This can be enabled by setting item 218 of the OPTIONS keyword to 2; in this case, for wells or well completions for which D-Factor calculations have been specified via the WDFAC keyword, the WDFACCOR keyword or item 12 of the COMPDAT keyword (or item 13 of the COMPDATL keyword), these will supersede those due to the VDFLOWR or VDFLOW keyword.

**Note:** For ECLIPSE 300, the non-Darcy flow calculation specified via the VDFLOW keyword (or the VDFLOWR keyword) is not applied to the well inflow calculations.

See also keyword VDFLOW.

**Example**

```
-- with NTSFUN=3
VDFLOWR
10.0 /
12.0 /
14.0 /
```

Keywords

Keywords

2372
Gas velocity dependent relative permeability data

The data comprises of a set of tables for the capillary number and Forchheimer models of gas velocity dependent relative permeability. The number of tables should be equal to the number of saturation tables in the run, specified by the TABDIMS keyword or defaulted to 1.

Each table consists of the following items of data, and is terminated with a slash (/).

For further details on this option, please refer to “Non-Darcy flow” in the ECLIPSE Technical Description. The capillary number and Forchheimer models for the velocity dependent relative permeabilities are set with the keyword VELDEP.

items in each table:

1. The \( m_g \) parameter.
   
   This parameter controls the variability of the critical gas saturation with normalized capillary number. The lower limit value of zero implies the critical gas saturation is always zero regardless of capillary number. A large positive value means the critical gas saturation hardly changes during the run.

2. The \( n_{1g} \) parameter.
   
   This parameter, along with \( n_{2g} \) parameter, controls the weighting between the miscible and immiscible relative permeability curves.

3. The \( n_{2g} \) parameter.
   
   This parameter along with \( n_{1g} \) parameter controls the weighting between the miscible and immiscible relative permeability curves.

4. The base capillary number for gas, \( N_{cbg} \)
   
   This is the threshold value of capillary number above which the VDRP effect is thought to be active.

5. The gas Forchheimer \( a_g \) parameter.
   
   This value must be specified for the \( B_1 \) Forchheimer model.

6. The gas Forchheimer \( b_g \) parameter.
   
   This value must be specified for the \( B_1 \) Forchheimer model.

7. The gas Forchheimer \( c_g \) parameter.

8. The gas Forchheimer \( d_g \) parameter.

9. The gas Forchheimer \( \beta_{dg} \) parameter.

   This value must be specified for the \( B_2 \) Forchheimer model.

All the above items default to 0.0. Warnings will be given if, for the particular models chosen in the VELDEP keyword, non-zero values are required.

UNITS: All items are dimensionless except for items 5 and 9 which have a unit which is a combination of the Forchheimer unit, Forch (where 9.86 Forch = 10^7 cm^-1), and the permeability unit (mD):

- item 5 has units of Forch(mD)^d
- item 9 has units of Forch/(mD)^d
where \( d \) is the \( d_g \) specified in item 8.

See also the VDKRG keyword and the VELDEP keyword in RUNSPEC.

**Example**

Gas capillary number and \( B_1 \) Forchheimer dependency have been requested using the VELDEP keyword in the RUNSPEC section. There are 2 saturation regions.

```
VDKRG
  -- M    N1    N2    Ncb    a    b    c    d    Betad
  50.0  3.0  -1.0  2.0E-06  8.0  0.0  0.0  0.5  1*    /
  35.0  3.5  -1.5  2.0E-06  9.0  0.0  0.0  0.5  1*    /
```
Gas condensate velocity dependent relative permeability data

The data comprises of a set of tables for a capillary number based model of velocity dependent relative permeability for near wellbore gas flow in a gas condensate reservoir. The number of tables should be equal to the number of saturation tables in the run, specified by the TABDIMS keyword or defaulted to 1.

This model is based on the work of Whitson, Fevang and Saevareid. For further details on this option, see "Non-Darcy flow" in the ECLIPSE Technical Description.

The data in this keyword are used in equations and . The capillary number and Forchheimer models for the velocity dependent relative permeabilities are set with the keyword VELDEP, with the gas condensate capillary number model activated by setting item 5 of this keyword to 1 (and item 2 to 0).

Each table consists of the following items of data, and is terminated with a slash (/):

1. The $n$ exponent.

   This exponent is used in the calculation of the transition function that combines the immiscible (user supplied) relative permeability curve with the miscible (straight line) relative permeability curve.

   UNITS: dimensionless
   DEFAULT: 0.65

2. The $\alpha_c^0$ coefficient.

   This coefficient is used in the capillary number scaling parameter.

   UNITS: dimensionless
   DEFAULT: 1.0E4

Example

There are 2 saturation regions.

```
VDKRGC
  n alpha0
  0.65 1.0e4 /
  0.70 1.5e4 /
```
Oil velocity dependent relative permeability data

The data comprises a set of tables for the capillary number and Forchheimer models of oil velocity dependent relative permeability data. The number of tables should be equal to the number of saturation tables in the run, specified by the `TABDIMS` keyword or defaulted to 1.

Each table consists of the following items of data, and is terminated with a slash (/).

For further details on this option, refer to "Non-Darcy flow" of the ECLIPSE Technical Description. The capillary number and Forchheimer models for the velocity dependent relative permeabilities are set with the keyword `VELDEP`.

Items in each table:

1. The \( m_o \) parameter.
   
   This parameter controls the variability of the critical oil saturation with normalized capillary number. The lower limit value of zero implies the critical oil saturation is always zero regardless of capillary number. A large positive value means the critical oil saturation hardly changes during the run.

2. The \( n_{1o} \) parameter.
   
   This parameter, along with the \( n_{2o} \) parameter, controls the weighting between the miscible and immiscible relative permeability curves.

3. The \( n_{2o} \) parameter.
   
   This parameter, along with the \( n_{1o} \) parameter, controls the weighting between the miscible and immiscible relative permeability curves.

4. The base capillary number for oil, \( N_{cbo} \).
   
   This is the threshold value of capillary number above which the VDRP effect is thought to be active.

5. The oil Forchheimer \( a_o \) parameter.
   
   This value must be specified for the B\( _1 \) Forchheimer model.

6. The oil Forchheimer \( b_o \) parameter.
   
   This value must be specified for the B\( _1 \) Forchheimer model.

7. The oil Forchheimer \( c_o \) parameter.

8. The oil Forchheimer \( d_o \) parameter.

9. The oil Forchheimer \( \beta_{do} \) parameter.
   
   This value must be specified for the B\( _2 \) Forchheimer Model.

All the above items default to 0.0. Warnings will be given if, for the particular models chosen in the `VELDEP` keyword, non-zero values are required.

UNITS: All items are dimensionless except for items 5 and 9 which have a unit which is a combination of the Forchheimer unit, Forch (where 9.86 Forch = 10\(^7\) cm\(^{-1}\)), and the permeability unit (mD):

- item 5 has units of Forch(mD)

---

**Keywords**

2376
• item 9 has units of Forch/(mD)^d

where d is the d_o specified in item 8.

See also the VDKRG keyword and the VELDEP keyword in RUNSPEC.

**Example**

Oil capillary number has been requested via the VELDEP keyword in RUNSPEC.

```
VDKRG -- M    N1   N2    Ncb    a    b    c    d    Betad
0.0  4.0  -1.5  2.0E-07  0.0  0.0  0.0  0.0  1*    /
```
### The vertical equilibrium model is used

This indicates that the vertical equilibrium model is required. The keyword may optionally be followed by a single item to activate the compressed VE model.

The data should be terminated by a slash (/).

1. **Type of VE model**
   - **COMPRESS**  The compressed vertical equilibrium model is used: columns of cells is compressed into a single cell.
   - **NOCOMP**  The standard VE model is to be used.

   **DEFAULT:** NOCOMP

See "Vertical equilibrium" in the *ECLIPSE Technical Description* for more details.

---

**CAUTION:** This keyword is banned if the COMPRESS option is used in a parallel run.

---

**Note:** The end-point scaling option/initial mobile fluid end point correction should not be used with the vertical equilibrium option.

---

### Example

The standard VE model is to be used

```
VE
NOCOMP /
```
VECTABLE | Size of vector property table

The data is an integer value, specifying the size of the integer table used for fast property lookups. A default size of 1000 is used, but if this does not allow an integer table to be initialized then ECLIPSE 300 suggests using the VECTABLE keyword and report the minimum integer that allows a fast lookup table to be generated.

Example

```plaintext
VECTABLE
2000 /
```
VEDEBUG Controls debug for the VE and compressed VE options

This keyword can be used to write out debug information (to the Debug file) about the internal tables used by the compressed VE option and the full (3D) VE option (keyword VE in the RUNSPEC section).

In the compressed VE case, the tables can be output in the form of a GRAF user file which can be read directly into the GRAF program.

The keyword should be followed by 7 integers, and optionally an LGR name, terminated with a slash (/). The first 6 integers define the box of cells for which the debug output will be generated. The 7th integer controls the form of the debug output.

1. IX1 First cell on X axis to be included in output
2. IX2 Last cell on X axis to be included in output
3. JY1 First cell on Y axis to be included in output
4. JY2 Last cell on Y axis to be included in output
5. KZ1 First cell on Z axis to be included in output
6. KZ2 Last cell on Z axis to be included in output

The values must satisfy:

1 \leq IX1 \leq IX2 \leq NDIVIX
1 \leq JY1 \leq JY2 \leq NDIVIY
1 \leq KZ1 \leq KZ2 \leq NDIVIZ (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS)

7. Debug switch
   0 Output in tabular form
   1 Output as GRAF user data

(Only if COMPRESS is specified in keyword VE
DEFAULT: 0

8. LGR control string
   ' ' Output cells in global grid and all local grids
   GLOBAL Output cells in global grid only
   lgr name Output cells in the named local grid

This item will be ignored if Local Grid Refinement is not used (see "Local grid refinement and coarsening" in the ECLIPSE Technical Description).
DEFAULT: ' '
Note: The output can be quite large, and hence only small numbers of cells should be output from large models.

Example

```
VEDEBUG
   -- IX1 IX2 IY1 IY2 IZ1 IZ2 SWITCH
      4   4   5   5   1   3     0     /
```
VEFIN may be used in LGR runs with vertical equilibrium in the global grid (keyword \texttt{VE} in the \texttt{RUNSPEC} section), to control the vertical equilibrium model in the local grid systems. VEFIN should be placed after the keyword introducing the local grid (\texttt{CARFIN} or \texttt{RADFIN}) and before the local grid data is terminated with \texttt{ENDFIN}.

The keyword can be used to specify that a local grid uses the vertical equilibrium model. In addition it allows you to control the accuracy of the global host cells’ representation for VE calculations, approximating the local grid systems.

The keyword is followed by one record containing two items of data, which relate to the local grid system introduced in the preceding \texttt{CARFIN} or \texttt{RADFIN} keyword:

1. Vertical Equilibrium flag:
   - \texttt{YES} If vertical equilibrium is required in the local grid.
   - \texttt{NO} To use dispersed flow in the local grid.

   DEFAULT: NO

2. \texttt{NVEPT} The maximum number of distinct depths used to describe a global grid cell hosting the local grid. (VE calculations use a table of depth versus volume for each cell.)

   If \texttt{NVEPT} is set to zero or negative, the original global description of the cell will be used (that is up to 8 distinct depths).

   If \texttt{NVEPT} is set to a large number, the global cell description honors the distinct depths of the local grid system. This means that the global cell contact depths are calculated using exactly the same data as the local grid system. (\texttt{NVEPT} needs to be set greater than 8 multiplied by the maximum number of local grid cells contained within any one global grid cell hosting the refinement.)

   If the number of local cells contained within any global cell in the refinement is large, the amount of computer time required to set up the initial data can become excessive, and the extra points will give no increase in accuracy. However, if \texttt{NVEPT} is set less than the number of distinct local grid depths within a global cell, then only \texttt{NVEPT} equally spaced distinct depths will be used. Note that if \texttt{NVEPT} is quite small (10 to 20), this is still a better approximation of the local system than the original (\texttt{NVEPT} = 0) description of the global cell.

   In general it is unlikely to be worth setting \texttt{NVEPT} to a value greater than 50.

   DEFAULT: 0 (which applies the original description of the global cell.)

\textit{Note:} The equilibration option selected by the item 9 in the \texttt{EQUIL} keyword is \textbf{not} affected by the VEFIN setting. If VE equilibration is requested (item 9 of \texttt{EQUIL} set to 0 in VE runs) then the local grid also uses VE equilibration. The VE equilibration option can be useful in cases with no capillary pressure where an accurate initial oil distribution is required; however, this is not in initial equilibrium. If a standard equilibration is required, which includes a transition zone if the capillary pressure is non-zero, then an equilibration region should be defined to cover the local refinement and item 9 of \texttt{EQUIL} should be set non-zero. Note that a value of 1 requests block-centered equilibration.
Examples

Example 1

Specify VE in the local grid.

```plaintext
VEFIN
YES 
```

Example 2

Increase the number of distinct depths used to represent the global host cell (with no VE in the local grid)

```plaintext
VEFIN
-- NVEPT
NO 40 
```

Keywords

2383
VEFRAC

Fraction of VE relative permeability curves used

If the vertical equilibrium option is being used (keyword VE in the RUNSPEC section), this keyword can be used to specify the relative fractions of the rock relative permeability curve and VE relative permeabilities to be used in the run.

The value of VEFRAC can either be specified as a single parameter for the whole field using this keyword, or it can be specified on a cell-by-cell basis using the VEFRACV keyword. The corresponding keywords VEFRACP and VEFRACPV can be used to specify the relative fractions of the rock and VE curves for capillary pressures.

DEFAULT: 1.0

If the default value is taken, relative permeabilities derived from the Vertical Equilibrium Model are used, replacing the rock curves entered using keywords such as SWPN.

By setting VEFRAC to a value less than 1.0, it is possible to mix the VE and rock curves; a fraction (1.0 - VEFRAC) of the rock curve and VEFRAC of the VE curve is taken.

See also keywords VEFRACV, VEFRACP, VEFRACPV.

Example

```plaintext
VEFRAC
0.9 /
```

Keywords

2384
VEFRACP  Fraction of VE pseudo capillary pressure to be used

If the vertical equilibrium option is being used (keyword `VE` in the `RUNSPEC` section), this keyword can be used to specify the relative fractions of rock capillary pressure curve and VE pseudo capillary pressure used in the run.

The value of `VEFRACP` can either be specified as a single parameter for the whole field using this keyword, or it can be specified on a cell-by-cell basis using the `VEFRACPV` keyword. The corresponding keywords `VEFRAC` and `VEFRACV` can be used to specify the relative fractions of the rock and VE curves for relative permeabilities.

**DEFAULT:** 1.0

If the default value is taken, capillary pressures derived from the vertical equilibrium model are used, replacing the rock curves entered using keywords such as `SWFN`.

By setting `VEFRACP` to a value less than 1.0, it is possible to mix the VE and rock curves; a fraction \((1.0 - \text{VEFRACP})\) of the rock curve and \(\text{VEFRACP}\) of the VE curve is taken.

See also keywords `VEFRAC`, `VEFRACV`, `VEFRACPV`.

**Example**

```
VEFRACP
0.7 /
```
VEFRACPV  Fraction of VE pseudo capillary pressure to be used

If the vertical equilibrium option is being used (keyword VE in the RUNSPEC section), this keyword can be used to specify the relative fractions of the rock capillary pressure curve and VE pseudo capillary pressure used in the run.

This keyword is used to input the VEFRAC parameter on a cell-by-cell basis.

DEFAULT: 1.0 (Throughout the entire grid.)

When VEFRAC is set to 1.0 (the default value), the capillary pressures derived from the vertical equilibrium model are used, replacing the rock curves entered using keywords such as SWFN.

By setting VEFRAC to a value less than 1.0, it is possible to mix the VE and rock curves; a fraction (1.0 - VEFRAC) of the rock curve and VEFRAC of the VE curve is taken.

The keyword can be used in conjunction with the EQUALS, ADD and MULTIPLY keywords.

See also keywords VEFRAC, VEFRACV, VEFRACP.

Example

```
--------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX       1   8   1   1   2   /
VEFRACPV  8*0.5
          8*1.0 /
```

Keywords

2386
VEFRACV  Fraction of VE relative permeability curves used

If the vertical equilibrium option is being used (keyword VE in the RUNSPEC section), this keyword can be used to specify the relative fractions of the rock relative permeability curve and VE relative permeabilities to be used in the run.

This keyword is used to input the VEFRAC parameter on a cell-by-cell basis.

DEFAULT: 1.0 (throughout the entire grid.)

When VEFRAC is set to 1.0 (the default value), the relative permeabilities derived from the vertical equilibrium model are used, replacing the rock curves entered using keywords such as SWFN.

By setting VEFRAC to a value less than 1.0, it is possible to mix the VE and rock curves; a fraction (1.0 - VEFRAC) of the rock curve and VEFRAC of the VE curve is taken.

The keyword can be used in conjunction with the EQUALS, ADD and MULTIPLY keywords.

See also keywords VEFRAC, VEFRACP, VEFRACPV.

Example

```
---------- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX       1 8 1 1 1 2 /
VEFRACV
8*0.5
8*1.0 /
```
**Velocity dependent relative permeability settings**

The data consists of the following items, describing the options available within the VDRP Model.

1. This item controls which Capillary Number Model is active for the **oil** phase. The 3 models available are selected by setting this item to 1, 2 or 3.
   
   **DEFAULT:** 0 (indicates this model is inactive)

2. This item controls which Capillary Number Model is active for the **gas** phase. The 3 models available are selected by setting this item to 1, 2 or 3.
   
   **DEFAULT:** 0 (indicates this model is inactive)

3. This item controls which Forchheimer Model is active for the **oil** phase. The 2 models available are selected by setting this item to 1 or 2.
   
   **DEFAULT:** 0 (indicates this model is inactive)

4. This item controls which Forchheimer Model is active for the **gas** phase. The 2 models available are selected by setting this item to 1 or 2.
   
   **DEFAULT:** 0 (indicates this model is inactive)

5. This item controls whether an alternative model for Capillary Number effects in near wellbore flows in gas condensate reservoirs is active for the **gas**, or gas and oil, phases. This model is activated for the gas phase only by setting this item to 1, and activated for gas and oil phases by setting this item to 2. When using this alternative model, item 2 must be set to zero.
   
   **DEFAULT:** 0 (indicates this model is inactive)

The description of the Velocity Dependent Relative Permeability (VDRP) model is discussed in "Non-Darcy flow" in the *ECLIPSE Technical Description*. The maximum deviation from non-Darcy flow is given in the REPORT section of the Print file, for example:

```
Maximum deviation from Darcy flow    .000259
```

This keyword does not work with black oil runs since the VDRP Model is intended for gas condensate systems.

Properties of the capillary number and Forchheimer models for the oil and gas are set with the keywords *VDKRO* and *VDKRG* respectively.

Properties of the alternative model for capillary number effects in near wellbore flow in gas condensate reservoirs have default values, but these values may be modified using *VDKRGC* keyword in the PROPS section.

VDRP effects (both capillary number and Forchheimer) may be modeled in the generalized pseudo-pressure calculation by setting item 4 to YES in the keyword *PICOND*.

**Example**

The $N_1$ models for oil and gas capillary number dependency are required along with the $B_1$ model or the Forchheimer effect applied to gas only.

```
VELDEP
  1 1 0 1 0 /
```

*Keywords*
VFP table consistency check data

This keyword is generated by VFP when it is working in its compositional mode. It contains data that was used by VFP to generate the VFPPROD and/or VFPINJ keywords. ECLIPSE 300 checks the data for consistency between the simulator fluid model and that used by VFP. A warning message is printed if there is an inconsistency.

The keyword is followed by a line containing the following items of data, terminated with a slash (/).

1. Water density at standard conditions
   UNITS: kg/m³ (METRIC), lb/ft³ (FIELD), gm/cc (LAB), kg/m³ (PVT-M)
2. Molecular weight of water
   UNITS: kg/kg-M (METRIC), lb/lb-M (FIELD), gm/gm-M (LAB), kg/kg-M (PVT-M)
3. Standard pressure
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
4. Standard temperature
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
5. Units system
6. Number of components

Example

```
VFPCHECK
1022.0 18.02 1.0132 15.5556 METRIC 15 /
```
Sets threshold BHP for VFP table checks

When production well VFP tables are read in, they are checked to ensure that curves of BHP versus flow rate at adjacent THP values do not cross. If a crossing is detected (that is if the BHP increases with decreasing THP) a warning message is printed. It may cause well convergence problems or hunting between control modes. However, crossings may also occur if some BHP values are set artificially high to indicate conditions where the flow is supersonic or exceeds specified erosion velocity limits (see keywords "SONICPRS" and "ERODEVEL" in the VFPi User Guide). These are legitimate, and serve to prevent the well from operating at these conditions. The VFP table check can be thresholded (to report only crossover BHPs below a given value), or even turned off entirely, with the keyword VFPCHK. The keyword should be entered before the VFPPROD keywords to which it should apply.

The VFPCHK keyword is followed by a single real number terminated with a slash (/), which sets the threshold BHP value for the checks. Any crossover BHPs at or above the threshold value will be ignored by the checks. A zero or negative value turns off the checks entirely.

**UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

**DEFAULT:** 1.0E10

If this keyword is not present, checks are performed with the default threshold BHP value.

---

**CAUTION:**

When ECLIPSE 300 encounters a crossing curve, it attempts to correct this internally by reducing the higher BHP value (on the lower THP curve) to lie below the lower BHP value (on the higher THP curve), so that the BHP increases with increasing THP. However, if the BHP values had been made artificially high on purpose to distinguish regimes of supersonic or erosional flow, the correction removes this distinction and allow the well to operate under these conditions. It is therefore important to use VFPCHK to prevent ECLIPSE 300 altering points that have been given artificially high BHP values for this purpose.

---

**Example**

```
VFPCHK
1.0E5 /
```
**Injection well VFP table dimensions**

The data consists of some or all of the following items which describe the dimensions of the injection well vertical flow performance tables entered in the *SCHEDULE* section using keyword `VFPINJ`. The data must be terminated by a slash (/).

1. The maximum number of flow values per table
   - DEFAULT (ECLIPSE 100): 0
   - DEFAULT (ECLIPSE 300): 1

2. The maximum number of tubing head pressure values per table
   - DEFAULT (ECLIPSE 100): 0
   - DEFAULT (ECLIPSE 300): 1

3. The maximum number of injection well VFP tables
   - DEFAULT (ECLIPSE 100): 0
   - DEFAULT (ECLIPSE 300): 1

**Example**

```
VFPIDIMS 10 2 5 /
```
VFPINJ

Inputs a VFP table for injection wells

This is a table of BHP versus FLO and versus THP.

At least one table is needed if any of the injection wells require THP calculations. The **VFPINJ** keyword can only enter one table at a time, so if there is more than one table, each table must be entered under a separate **VFPINJ** keyword.

See also keyword **WVFPDP**.

The maximum dimensions of the tables must be set in the **RUNSPEC** section keyword **VFPIDIMS**.

The table consists of the following records of data, each record terminated with a slash (/).

**Record 1: Basic data for table**

The last three items may be defaulted, for back-compatibility with earlier versions.

1. **Table number**
   
   In integer between 1 and NMSVFT, the maximum number of injection well VFP tables, which is the third item in **VFPIDIMS**.

2. **Bottom hole datum depth for table.**
   
   Any difference between the bottom hole datum depth of the VFP table and the bottom hole datum depth of the wells that use the table is taken into account by a simple hydrostatic correction.
   
   This item is ignored if the table is used to determine branch pressure drops in the Network option.
   
   **UNITS:** m (**METRIC**), ft (**FIELD**), cm (**LAB**), m (**PVT-M**)

3. **Definition of the flow rate variable FLO.**
   
   - **OIL**  oil injection rate
   - **WAT**  water injection rate
   - **GAS**  gas injection rate
   
   **ECLIPSE 300 only**
   - **WG**  hydrocarbon wet gas rate
   
   **ECLIPSE 300 only**
   - **TM**  total molar rate

4. **Definition of the fixed pressure values given in "Record 3:THP values".**

   **ECLIPSE 300 only**
   - **THP**  tubing head pressure
   
   This is interpreted as the inlet pressure if the table is used to determine injection network branch pressure drops in the Network option.
   
   **ECLIPSE** does not accept anything other than **THP**.
   
   **DEFAULT:**  **THP**

---

**Keywords**

2392
5. Unit convention of the VFP table.
   METRIC, FIELD, LAB or PVT-M.
   If this item is present, ECLIPSE checks that the VFP table has the same unit convention as the run and
   issues an error message if they are not the same.
   DEFAULT: The same unit convention as the ECLIPSE run.

6. Definition of the tabulated quantity in the body of the table, records 4 onwards.
   BHP Bottom hole pressure
   This is interpreted as the outlet pressure if the table is used to determine injection network branch
   pressure drops in the Network option.
   ECLIPSE will not accept anything other than BHP.
   DEFAULT: BHP

**Record 2:FLO (flow rate) values**

Enter in ascending order, terminated with a slash (/).

The number of values, NFLO, must be between 2 and MXSFLO, which is the first item in VFPIDIMS.

Volume rate: UNITS: sm$^3$/day (METRIC), stb/day for oil or liquid (FIELD), Mscf/day for gas (FIELD),
sec/hr (LAB), sm$^3$/day (PVT-M)

Molar rate: UNITS: kg-M/day (METRIC), lb-M/day (FIELD), gm-M/hr (LAB), kg-M/day (PVT-M)

**Record 3:THP values**

Enter in ascending order, terminated with a slash (/).

The number of values, NTHP, must be between 1 and MXSTHP, which is the second item in VFPIDIMS. If
only one value is entered, the derivative of the BHP with respect to the THP is treated as unity.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

**Records 4 to NTHP+3**

NTHP successive records each containing the following items, and each terminated with a slash (/).

1. NT (THP value number).
2. BHP at 1$^{st}$ FLO value and NT$^{th}$ THP value.
3. BHP at 2$^{nd}$ FLO value and NT$^{th}$ THP value.
   ...
   ...

NFLO +1 BHP at last FLO value and NT$^{th}$ THP value.

All combinations of NF = 1 to NFLO and NT = 1 to NTHP must be covered.
UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
### Examples

#### Example 1

VFP table for water injector, NFLO=5, NTHP=1:

```plaintext
VFPINJ
1 7.00000E+03 WAT THP FIELD BHP / 1.00000E+00 3.00000E+02 7.00000E+02 1.00000E+03 2.00000E+03 / 1.00000E+03 / 1 4.03999E+03 4.03423E+03 4.01346E+03 3.98903E+03 3.85537E+03 /
```

#### Example 2

VFP table for gas injector, NFLO=5, NTHP=3:

```plaintext
VFPINJ
2 7.00000E+03 GAS THP FIELD BHP / 1.00000E+00 3.00000E+02 7.00000E+02 1.00000E+03 2.00000E+03 / 1.00000E+03 2.00000E+03 3.00000E+03 / 1 1.32484E+03 1.32300E+03 1.31556E+03 1.30626E+03 1.25098E+03 / 2 2.74881E+03 2.74801E+03 2.74490E+03 2.74110E+03 2.71934E+03 / 3 3.94062E+03 3.94000E+03 3.93761E+03 3.93471E+03 3.91830E+03 /
```
Production well VFP table dimensions

The data consists of some or all of the following items which describe the dimensions of the production well vertical flow performance tables entered in the SCHEDULE section using keyword `VFPPROD`. The data must be terminated by a slash (/).

1. The maximum number of flow values per table
   - DEFAULT (ECLIPSE 100): 0
   - DEFAULT (ECLIPSE 300): 1

2. The maximum number of tubing head pressure values per table
   - DEFAULT (ECLIPSE 100): 0
   - DEFAULT (ECLIPSE 300): 1

3. The maximum number of water fraction values per table
   - DEFAULT (ECLIPSE 100): 0
   - DEFAULT (ECLIPSE 300): 1

4. The maximum number of gas fraction values per table
   - DEFAULT (ECLIPSE 100): 0
   - DEFAULT (ECLIPSE 300): 1

5. The maximum number of Artificial Lift Quantities per table
   - DEFAULT (ECLIPSE 100): 0
   - DEFAULT (ECLIPSE 300): 1

6. The maximum number of production well VFP tables
   - DEFAULT (ECLIPSE 100): 0
   - DEFAULT (ECLIPSE 300): 1

Note: Production well VFP tables can use a significant amount of computer storage, as the reserved memory is proportional to the product of all the above numbers. It is therefore good practice to keep these maximum dimensions as close as possible to the actual maximum dimensions of the tables entered.

To help select the most appropriate values, if the mnemonic 'VFPPROD=2', or higher, is entered in the `RPTSCHED` keyword then a table is produced comparing the actual maximum dimensions of the VFP tables with the dimensions input here. Enter 'VFPPROD=3' if you do not wish to have the contents of the VFP tables printed as well.

Example

```
VFPPDIMS
5 4 3 1 1 3 /
```
VFPPROD  Inputs a VFP table for production wells

This is a table of BHP versus FLO, THP, WFR, GFR and ALQ, or THT (Tubing Head Temperature) versus FLO, THP, WFR, GFR and ALQ, where:

- **FLO** is the oil, liquid or gas production rate, or, in compositional mode, the hydrocarbon wet gas production or total molar rate.
- **WFR** is the water-oil ratio, water cut or water-gas ratio, or, in compositional mode, the water-wet gas ratio or water molar fraction.
- **GFR** is the gas-oil ratio, gas-liquid ratio or oil-gas ratio, or, in compositional mode, the hydrocarbon mean molecular weight.
- **ALQ** is a 5th variable that can be used to incorporate an additional look-up parameter, such as the level of artificial lift.

Any combination of these variable definitions can be used, as long as they do not become infinite during the simulation.

For example, an oil well may have

- **FLO** oil or liquid rate, or, in compositional mode, the wet gas or molar rate.
- **WFR** water-oil ratio or water cut, or, in compositional mode, the water-wet gas ratio or water molar fraction.
- **GFR** gas-oil ratio or gas-liquid ratio, or, in compositional mode, the mean molecular weight.

A gas well should have

- **FLO** gas rate, or, in compositional mode, the wet gas or molar rate.
- **WFR** water-gas ratio, or, in compositional mode, the water-wet gas ratio or water molar fraction.
- **GFR** oil-gas ratio (in gas-water runs only one value = 0.0 is required).

**ECLIPSE 100**

If the **RSCONST** keyword is used, the variables must be

- **FLO** oil or liquid rate
- **WFR** water-oil ratio or water cut
- **GFR** gas-oil ratio.

**ECLIPSE 100**

If the **RVCONST** keyword is used, the variables must be

- **FLO** gas rate
- **WFR** water-gas ratio
- **GFR** oil-gas ratio.
At least one table is needed if any of the production wells require THP calculations. The `VFPPROD` keyword can only enter one table at a time, so if there is more than one table, each table must be entered under a separate `VFPPROD` keyword.

**Note:** If the actual simulation conditions fall outside the range given in this table, linear extrapolation is used. This can produce unrealistic results and should be avoided (by ensuring that the VFP table covers all conditions that will arise during the simulation).

The maximum dimensions of the tables must be set in the `RUNSPEC` section keyword `VFPPDIMS`.

See also keywords `VFPTABL, VFPCHK, WVFDP, WVFPEXP` and `WHTEMP`.

The table consists of seven records of data, each record terminated with a slash (/).

### Record 1: Basic data for table

The last four items may be defaulted, for back-compatibility with earlier versions.

1. **Table number**
   
   An integer between 1 and `NMMVFT`, the maximum number of production VFP tables, which is item 6 in `VFPPDIMS`.

   Each production VFP table must have a unique table number irrespective of whether the body of the table holds pressures or temperatures.

2. **Bottom hole datum depth for table.**

   Any difference between the bottom hole datum depth of the VFP table and the bottom hole datum depth of the wells that use the table are taken into account by a simple hydrostatic correction.

   This item is ignored if the table is used to determine branch pressure drops in the network option.

   If the table is used to determine segment pressure drops in the multisegment well model, this item is interpreted as either the length or the depth span of the representative length of tubing for which the table was calculated.

   **UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

3. **Definition of the flow rate variable FLO in record 2.**

   - **OIL** Oil production rate
   - **LIQ** Liquid (oil + water) production rate
   - **GAS** Gas production rate

   **ECLIPSE 300 only**
   - **WG** Hydrocarbon wet gas rate

   **ECLIPSE 300 only**
   - **TM** Total molar rate

4. **Definition of the water fraction variable WFR in record 4.**

   - **WOR** Water-oil ratio
   - **WCT** Water cut (water-liquid ratio)
WGR  Water-gas ratio

ECLIPSE 300 only

WWR  Water-wet gas ratio

ECLIPSE 300 only

WTF  Water molar fraction

ECLIPSE 300

Note: Water-wet gas ratio is defined as \( \frac{f_w^{\text{vol}}}{f_h^{\text{vol}}} \), where \( f_w^{\text{vol}} \) is the volume flow rate of water at surface conditions, and \( f_h^{\text{vol}} \) is the molar rate of hydrocarbon flow, converted to a gas volume rate at standard conditions.

Water molar fraction is defined as \( \frac{f_w^{\text{m}}}{f_h^{\text{m}} + f_w^{\text{m}}} \), where \( f_w^{\text{m}} \) is the molar flow rate of water at surface conditions, and \( f_h^{\text{m}} \) is the molar rate of hydrocarbon flow.

5. Definition of the gas fraction variable GFR in record 5.

GOR  Gas-oil ratio

GLR  Gas-liquid ratio

OGR  Oil-gas ratio

ECLIPSE 300 only

MMW  Mean molecular weight

6. Definition of the fixed pressure values given in record 3.

THP  Tubing head pressure

This is interpreted as the outlet pressure if the table is used to determine production network branch pressure drops in the network option, or segment pressure drops in the multisegment well model.

ECLIPSE will not accept anything other than THP.

DEFAULT: THP

7. Definition of the artificial lift quantity (ALQ) in record 6

GRAT  Lift gas injection rate (for gas lift)

IGLR  Injection gas-liquid ratio (for gas lift)

TGLR  Total gas-liquid ratio (for gas lift)

PUMP  Pump rating (for pump)

COMP  Compressor power (for compressor)

ECLIPSE 100 only

DENO  Surface density of oil (See keyword WALQCALC)

ECLIPSE 100 only

DENG  Surface density of gas (See keyword WALQCALC)

BEAN  Choke diameter (multisegment wells)
This item informs ECLIPSE of the ALQ definition that was used to generate the VFP table (for example by VFP $i$). However, in general ECLIPSE uses the ALQ only as a look-up parameter to interpolate the table, and attaches no physical significance to its definition. An exception to this is when gas lift is the artificial lift method and is used in conjunction with any one of the following options:

- The gas lift optimization facility
- Limits to the total lift gas applied to a group (keyword GLIFTLIM)
- Including lift gas flows in the pressure loss calculations of network branches (network option keyword GRUPNET item 6 or NODEPROP item 4).

In these cases, the ALQ must be GRAT, because ECLIPSE adds the well ALQs to give the group lift gas rates. ALQs are not additive in this respect if they refer to IGLR or TGLR. The VFP $i$ program may be used to convert VFP tables constructed with the IGLR or TGLR definition into tables where the ALQ is GRAT. If the ALQ is undefined, ECLIPSE assumes it is GRAT, and an error message is issued if it is defined as IGLR or TGLR.

**Note:** In the presence of artificial lift gas, the produced gas reported (for example using the FGPR, GGPR and WGPR summary keywords) does not include produced lift gas, only produced formation gas.

Another use of the ALQ is to represent the oil or gas surface density. VFP $i$ can produce a table spanning a range of oil or gas surface densities, and ECLIPSE interpolates it at the current surface density. This is useful in API tracking runs, and also in the Network option when wells in different PVT regions produce into the network. See the keyword WALQCRL for information.

DEFAULT: ' ' (none or undefined)

8. Unit convention of the VFP table.

METRIC, FIELD, LAB or PVT-M

If this item is present, ECLIPSE checks that the VFP table has the same unit convention as the run and issues an error message if they are not the same.

DEFAULT: The same unit convention as the ECLIPSE run

9. Definition of the tabulated quantity in the body of the table, record 7 onwards.

TEMP  Tubing Head Temperature (THT)

BHP  Bottom hole pressure

A BHP is interpreted as the inlet pressure if the table is used to determine production network branch pressure drops in the Network option, or segment pressure drops in the multisegment well model. ECLIPSE will not accept anything other than BHP or TEMP.

DEFAULT: BHP
**Record 2: FLO (flow rate) values**

Enter in ascending order, terminated with a slash (/).

The number of values, NFLO, must be between 2 and MXMFL0, which is item 1 in VFPPDIMS.

Volume rate: UNITS: \( \text{sm}^3/\text{day} \) (METRIC), \( \text{stb/day} \) for oil or liquid (FIELD), \( \text{Mscf/day} \) for gas (FIELD), \( \text{scc/hr} \) (LAB), \( \text{sm}^3/\text{day} \) (PVT-M)

Molar rate: UNITS: \( \text{kg-M}/\text{day} \) (METRIC), \( \text{lb-M/day} \) (FIELD), \( \text{gm-M/hr} \) (LAB), \( \text{kg-M/day} \) (PVT-M)

**Record 3: THP values**

Enter in ascending order, terminated with a slash (/).

The number of values, NTHP, must be between 1 and MXMTHP, which is item 2 in VFPPDIMS. If only one value is entered, the derivative of the BHP w.r.t. the THP is treated as unity.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

**Record 4: WFR values**

Enter in ascending order, terminated with a slash (/).

The number of values, NWFR, must be between 1 and MXMWFR, which is item 3 in VFPPDIMS. If only one value is entered, the BHP is treated as independent of the water fraction.

For WOR or WCT: UNITS: \( \text{sm}^3/\text{sm}^3 \) (METRIC), \( \text{stb/stb} \) (FIELD), \( \text{scc/scc} \) (LAB), \( \text{sm}^3/\text{sm}^3 \) (PVT-M)

For WGR or WWR: UNITS: \( \text{sm}^3/\text{sm}^3 \) (METRIC), \( \text{stb/Mscf} \) (FIELD), \( \text{scc/scc} \) (LAB), \( \text{sm}^3/\text{sm}^3 \) (PVT-M)

For WTF: UNITS: dimensionless

**Record 5: GFR values**

Enter in ascending order, terminated with a slash (/).

The number of values, NGFR, must be between 1 and MXMGFR, which is item 4 in VFPPDIMS. If only one value is entered, the BHP is treated as independent of the gas fraction.

For GOR or GLR: UNITS: \( \text{sm}^3/\text{sm}^3 \) (METRIC), \( \text{Mscf/stb} \) (FIELD), \( \text{scc/scc} \) (LAB), \( \text{sm}^3/\text{sm}^3 \) (PVT-M)

For OGR: UNITS: \( \text{sm}^3/\text{sm}^3 \) (METRIC), \( \text{stb/Mscf} \) (FIELD), \( \text{scc/scc} \) (LAB), \( \text{sm}^3/\text{sm}^3 \) (PVT-M)

For MMW: UNITS: dimensionless

**Record 6: ALQ values**

Enter in ascending order, terminated with a slash (/).

The number of values, NALQ, must be between 1 and MXMALQ, which is item 5 in VFPPDIMS. If only one value is entered, the BHP is treated as independent of the artificial lift quantity.
When ALQ = GRAT: 
UNITs: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M) 

When ALQ = IGLR or TGLR: 
UNITs: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M) 

When ALQ = DENO or DENG: 
UNITs: kg/m$^3$ (METRIC), lb/ft$^3$ (FIELD), gm/cc (LAB), kg/m$^3$ (PVT-M) 

When ALQ = BEAN: 
UNITs: mm (METRIC), 1/64ths in (FIELD) 

The units for PUMP or COMP depend on the definition of the pump rating or compressor power used in the construction of the VFP table. 

(See item 7 of Record 1.) 

**Records 7 to NTHP*NWFR*NGFR*NALQ+6** 

NTHP * NWFR * NGFR * NALQ successive records each containing the following items, and each terminated with a slash (/). 

1. NT (THP value number). 
2. NW (WFR value number) 
3. NG (GFR value number) 
4. NA (ALQ value number) 
5. BHP or THT at 1st FLO value 
   NT$^\text{th}$ THP value 
   NW$^\text{th}$ WFR value 
   NG$^\text{th}$ GFR value 
   NA$^\text{th}$ ALQ value 
6. BHP or THT at 2nd FLO value 
   NT$^\text{th}$ THP value 
   NW$^\text{th}$ WFR value 
   NG$^\text{th}$ GFR value 
   NA$^\text{th}$ ALQ value 
   ........................................
   NFLO+4 BHP or THT at last FLO value
NT<sup>th</sup> THP value
NW<sup>th</sup> WFR value
NG<sup>th</sup> GFR value
NA<sup>th</sup> ALQ value

All combinations of
NF = 1 to NFLO
NT = 1 to NTHP
NW = 1 to NWFR
NG = 1 to NGFR
NA = 1 to NALQ must be covered.

For THT:
UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

For BHP:
UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

VFP tables created by the VFP<sub>i</sub> program may contain some BHP values set to 1.0E10. This value indicates that the particular combination of flowing conditions is physically impossible, as the fluid in the well bore would exceed the sonic velocity. This may occur, for example, at high flow rates and GOR values and low THP values. The artificially high BHP ensures that during the simulation the well flowing conditions will not enter the forbidden region of unphysical combinations of flowing conditions.

However, because ECLIPSE interpolates linearly between adjacent points in the table, the 1.0E10 values influence the interpolated BHP all the way out to the “physical” table points adjacent to the forbidden region. The sharp change in BHP around these points may cause the wells to behave in an unexpected manner. You are therefore recommended to change any 1.0E10 values in the table to a pressure slightly greater than the reservoir pressure. This should reduce the sharp change in interpolated BHP, while still preventing the well conditions from entering the “forbidden region”. If ECLIPSE detects any 1.0E10 values when reading the VFP<sub>PROD</sub> data, it prints a warning message inviting you to replace them with a lower pressure.

This can be avoided by instructing VFP<sub>i</sub> to use another pressure value to indicate supersonic flow (see the "VFP<sub>i</sub> User Guide", and keyword SONICPRS in the VFP<sub>i</sub> manual). A suitable value would be a round number (therefore easily recognizable) that is slightly greater than the maximum expected reservoir pressure.

If a pressure of less than 1.0E10 is used to indicate supersonic flow, you may get a warning message when the table is read saying that the BHP increases with decreasing THP. In this case, you may use the VFP<sub>CHK</sub> keyword to set the threshold BHP to the value used to indicate supersonic flow, to prevent these values from triggering the warning.

**CAUTION:**

When ECLIPSE 300 encounters a crossing curve, it attempts to correct this internally by reducing the higher BHP value (on the lower THP curve) to lie below the lower BHP value (on the higher THP curve),
so that the BHP increases with increasing THP. However, if the BHP values had been made artificially high on purpose to distinguish regimes of supersonic or erosional flow, the correction removes this distinction and allow the well to operate under these conditions. It is therefore important to use VFPCHK to prevent ECLIPSE 300 altering points that have been given artificially high BHP values for this purpose.

The VFP $i$ program can be used to transform existing VFP tables in a number of ways. It can replace any 1.0E10 values with a more reasonable user-defined value. It can transform the ALQ definitions for gas lift into the additive GRAT definition. It can remove the unstable side of the VFP curves (to the left of the minimum), replacing it with lines of constant BHP, which may be useful in curing simulator problems caused by wells suddenly dying (see keyword $\text{WVFPEXP}$). It also contains extensive graphical facilities for examining and editing VFP tables in cross-section and displaying them in 3D.

Example

VFP table for producer, $\text{NFLO}=5$, $\text{NTHP}=2$, $\text{NWFR}=1$, $\text{NGFR}=3$, $\text{NALQ}=1$:

```
VFP_PROD:
  1  7.00000E+03  LIQ  WCT  GOR  THP  ' '  FIELD  BHP  / Basic data
  1.00000E+01 3.00000E+02 7.00000E+02 1.00000E+03
  2.00000E+02 1.00000E+03 / 2 THP values
  0.0 / 1 WFR value
  1.00000E+00 2.00000E+00 4.00000E+00 / 3 GFR values
  0.0 / 1 ALQ value
  1 1 1 1.93199E+03 1.36585E+03 6.77031E+02 7.15261E+02
   8.62436E+02 /
  2 1 1 2.73663E+03 2.73303E+03 2.75085E+03 2.77323E+03
   2.90209E+03 /
  1 1 2 1.77471E+03 4.33035E+02 5.38422E+02 6.30479E+02
    9.39472E+02 /
  2 1 2 2.51228E+03 2.38072E+03 2.35995E+03 2.26536E+03
    2.28849E+03 /
  1 1 3 1.64735E+03 4.41989E+02 6.95286E+02 8.81634E+02
    1.41797E+03 /
  2 1 3 1.46600E+03 1.78161E+03 1.80525E+03 1.85156E+03
    2.04484E+03 /
```
VFPTABL

Set method of VFP table interpolation for ALQ

By default, VFP tables are interpolated linearly in all variables (FLOW, THP, WFR, GFR, ALQ). However, there is an option to apply cubic spline interpolation for the ALQ (Artificial Lift Quantity) variable. This option is primarily intended to be useful for the gas lift optimization facility (see "Gas lift optimization" in the ECLIPSE Technical Description), in which the gradient of the wells’ performance with reference to the lift gas injection rate is important for deciding which wells could gain the most benefit from an increased lift gas supply.

With linear interpolation, the gradient of the wells’ performance with respect to the ALQ value can change abruptly at each tabulated ALQ point. This may result in the optimal lift gas allocation rate to each well being biased towards the tabulated ALQ values in the VFP table. Interpolation based on cubic splines causes the gradient to change more smoothly over the table, while still honoring the values at the tabulated points.

The VFPTABL keyword is followed by a single integer (either 1 or 2), terminated with a slash (/).

1. Use linear interpolation for all variables in the VFP tables.
2. Use cubic spline interpolation for the ALQ variable in the production well VFP tables, and linear interpolation for the other variables. If there are more than 20 ALQ values in a table, linear interpolation is used for this table regardless of the method requested.

If this keyword is not present, linear interpolation is used for all variables.

Example

```
VFPTABL
  2 /
```
**VISAGE**

**Activate VISAGE geomechanics workflow**

This keyword activates functionality for the VISAGE geomechanics workflow.

This workflow is described in the "Geomechanics external to ECLIPSE" section of the ECLIPSE Technical Description.

This keyword has no associated data.

<table>
<thead>
<tr>
<th>Keywords</th>
</tr>
</thead>
<tbody>
<tr>
<td>ECLIPSE 100</td>
</tr>
<tr>
<td>ECLIPSE 300</td>
</tr>
<tr>
<td>SPECIAL</td>
</tr>
<tr>
<td>RUNSPEC</td>
</tr>
<tr>
<td>GRID</td>
</tr>
<tr>
<td>EDIT</td>
</tr>
<tr>
<td>PROPS</td>
</tr>
<tr>
<td>REGIONS</td>
</tr>
<tr>
<td>SOLUTION</td>
</tr>
<tr>
<td>SUMMARY</td>
</tr>
<tr>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>
**VISCAQA**

Specify coefficients of Ezrokhi’s method for aqua viscosity

This keyword enables the default Ezrokhi coefficients for the aqua viscosity calculation to be overwritten. The data consists of a table composed of a series of 3 coefficients for each component which may dissolve in the aqua phase. In GASSOL, these components are specified using the SOLUAQA table; in CO2STORE, GASWAT and thermal KWTABTn, Ezrokhi’s coefficients are specified for all components, although it should be noted that the coefficients for component H2O are automatically taken to be zero, irrespective of alternative user-supplied inputs. This keyword associates a contribution of each dissolved component into the aqua viscosity calculation. See also “GASSOL solution in the aqueous phase” in the ECLIPSE Technical Description.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

This keyword is thus compatible with the CO2STORE, GASWAT, GASSOL and KWTABTn (THERMAL) options.

**Examples**

**Example 1**

Input data for a four-component study:

```
VISCAQA
2.42E-7 0.0 1.20E-9
2.22E-7 0.0 1.11E-9
2.00E-7 0.0 1.01E-9
2.01E-7 0.0 1.02E-9
/
```

**Example 2**

With two reservoir EoS regions and four-component study:

```
VISCAQA
2.42E-7 0.0 1.20E-9
2.22E-7 0.0 1.11E-9
2.00E-7 0.0 1.01E-9
2.01E-7 0.0 1.02E-9
/
2.42E-7 0.01 1.20E-9
2.22E-7 0.02 1.11E-9
2.00E-7 0.03 1.01E-9
2.01E-7 0.04 1.02E-9
```
Activates the viscous displacement option

This keyword requests that the viscous displacement mechanism should be modeled in dual porosity runs. If this option is used, the average size of the matrix blocks should be entered with the \texttt{LX}, \texttt{LY} and \texttt{LZ} keywords in the \texttt{GRID} section. \texttt{VISCD} is not available with the \texttt{PARALLEL} keyword.

See "Viscous displacement (ECLIPSE 100)" in the \textit{ECLIPSE Technical Description} for further information.

The keyword has no associated data.
**Viscref**

**Reference conditions for the viscosity / temperature tables**

This keyword should only be used for runs with the ECLIPSE 100 Temperature option (see keyword **TEMP**) or the ECLIPSE 300 Thermal option (see keyword **THERMAL**). With the ECLIPSE 100 Temperature option it is required when the **OILVISCT** or **WATVISCT** keyword is present. With the ECLIPSE 300 Thermal option the keyword is optional, and is only used if pressure dependent oil or water viscosities are required.

The data comprises **NTPVT** (see keyword **TABDIMS**) records describing the reference conditions for the viscosity values in the viscosity versus temperature tables. Each record consists of some or all of the following items of data terminated by a slash (/).

1. The reference pressure
   
   **UNITs:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
   
2. The reference Rs value. ECLIPSE 100 only.
   
   This data item is only required if the model contains dissolved gas (see keyword **DISGAS**).
   
   **UNITs:** sm³/sm³ (METRIC), Mscf/stb (FIELD), scc/scc (LAB)
   
3. The reference API gravity. ECLIPSE 100 only.
   
   This data item is only required if the API Tracking option is active (see keyword **API**).
   
   **UNITs:** °API

**Example**

With NTPVT=2:

```plaintext
VISCREF
3000 1.24 /
3000 1.24 /
```

**Keywords**

2408
This keyword is part of the VISAGE geomechanics workflow. This workflow is activated by the presence of the `VISAGE` keyword in the RUNSPEC section.

This workflow is described in the "Geomechanics external to ECLIPSE" section of the ECLIPSE Technical Description. In this workflow, external calculations are performed using the ECLIPSE fluid simulation results to compute the geomechanical response of the reservoir to changes in pressure, fluid saturations/composition and temperature. These external calculations are performed at particular report steps in the simulation, these report steps are known as stress steps.

The keyword should be followed by a list of report dates at which stress step calculations are required.

Each stress step date must be on a separate line terminated by a slash (/). The data is terminated by a null record (a slash on a line by itself).

A date consists of the following items of data:

1. The day of the month
   An integer between 1 and 31.
2. The name of the month abbreviated to three characters
   JAN, FEB, MAR, APR, MAY, JUN, JLY, AUG, SEP, OCT, NOV or DEC.
   JUL is an acceptable alternative to JLY.
3. The year
   A positive 4 digit integer.
4. The time
   A string of the format HH:MM:SS.SSSS
   DEFAULT: 00:00:00

**Note:** If a stress step date does not correspond to a report date then it will have no effect. Duplicate dates are ignored (that is, the simulator will act as though that particular stress step date had only been specified once). Any stress step dates which are specified as being before the simulation start date are ignored.

**Note:** If this keyword is present more than once, the combined list of dates specified in all the `VISDATES` keywords is treated as the stress step dates for the simulation.

**Example**

```
VISDATES
1 FEB 1985 /
1 MAR 1985 11:15:00 /
1 MAY 1985 /
1 JAN 1986 11:15:30.3333 /
1 JAN 1987 /
```

Keywords
This keyword specifies options for use in the VISAGE geomechanics workflow. This workflow is activated by the presence of the VISAGE keyword in the RUNSPEC section.

This workflow is described in the "Geomechanics external to ECLIPSE" section of the ECLIPSE Technical Description. In this workflow, external calculations are performed using the ECLIPSE fluid simulation results to compute the geomechanical response of the reservoir to changes in pressure, fluid saturations/composition and temperature. These external calculations are performed by particular report steps in the simulations, these report steps are known as stress steps.

The data consists of some or all of the following items. The data must be terminated by a slash (/).

1. Whether this is a VISAGE workflow initialization run.
   - YES  GRID, INIT and initial restart data are output, and the run is switched to NOSIM with the report steps that would be performed written to the EDESC communication file.
   - NO  Output of GRID, INIT and initial restart data is not required, and only stress steps are written to the VDESC communication file.
   DEFAULT: NO

2. Whether the run is to exit at the next stress step encountered.
   - YES  Exit the simulation when the next stress step is encountered.
   - NO  Continue the simulation when the next stress step is encountered.
   DEFAULT: NO

3. Whether the dynamic stress step mechanism is active.
   - YES  If the field pressure has changed by more than the relative tolerance in item 4 since the last stress step, then the next report step will be treated as a stress step.
   - NO  There are no dynamic stress steps.
   DEFAULT: NO

4. The relative tolerance on the field pressure for a dynamic stress step. This must not be a negative value.

   When the change in field pressure since the previous stress step (or since the beginning of the simulation if there have been no stress steps) divided by the pressure at the previous stress step (or initial pressure if there have been no stress steps) exceeds this tolerance, a dynamic stress step will be triggered as long as item 3 of this keyword is set to ‘YES’.

   DEFAULT: 0.05

5. Not used in this release

6. Whether the existing restart frequency is to be maintained.
   - YES  If there are RPT_RST requests to reduce the frequency of the restart files, then these should be honored in the current run.
NO The VISAGE workflow default restart frequency (RPTRST argument BASIC=2) should be used.

DEFAULT: NO

7. Whether the existing restart content is to be maintained.

YES If there are additional RPTRST contents over and above those required for the VISAGE workflow (mnemonics BASIC=2 and ROCKC), then these should be honored in the current run.

NO Only the content required for the VISAGE workflow should be output to the restart files.

DEFAULT: NO

8. Whether it is an error or a warning to not re-specify well connections (which are required in order to recompute well connection factors after a change of permeability) before the first timestep when performing a restart.

ERROR (or E) It is an error, and no simulation will be performed.

WARN (or W) It is a warning, and simulation will proceed using the well connection factors from the restart file.

DEFAULT: ERROR

Example

The example below is not a VISAGE workflow initialization run and it will stop when it next encounters a stress step. It has the dynamic stress step mechanism active and it will treat a report step as a stress step if the relative change in the field pressure since the last stress step is greater than 0.1 (that is 10%). Any user inputs for restart frequency and content will be honored in this run. If well connections have not been re-specified before the first timestep, it will produce an error.

```
VISOPTS
  NO YES YES 0.10 1* YES YES ERROR/
```
VREFW

Reference viscosity and viscosibility for water components

VREFW specifies the water viscosity and viscosibility at a reference pressure for each of the \( N_w \) water components.

The water component viscosities, at pressure \( P \), are calculated from:

\[
\mu_w = \mu_{\text{ref}w} \left( 1 + Y + \frac{Y^2}{2} \right)
\]

\[
Y = -c_{Vw} \cdot (P - P_{\text{ref}w})
\]

where

\( \mu_{\text{ref}w} \) is the reference viscosity, specified with this keyword

\( c_{Vw} \) is the viscosibility, specified with this keyword

\( P_{\text{ref}w} \) is the reference pressure, specified with keyword \textsc{prefw}

The keyword is followed by 2 x \( N_w \) values.

1. The first \( N_w \) values represent the reference viscosity for each water component:

   \text{UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)}

   \text{DEFAULT: 0.3 cP.}

2. The second \( N_w \) values represent the viscosibility at the reference pressure for each water component:

   \text{UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)}

   \text{DEFAULT: 0.0.}

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered in item 9 of keyword \textsc{tabdims}. The use of multiple EoS regions is discussed in "Equations of state regions" in the \textit{ECLIPSE Technical Description}. See "Water components" in the \textit{ECLIPSE Technical Description} for further details of the water component option.

Example

```
VREFW 0.31 0.32
   0.0   0.0 /
```

Keywords

2412
This section contains an alphabetic listing of keywords starting with the letter W.
WADVANCE  Assign advance import gas to a well

This keyword assigns a supply of advance import gas to a well supplying gas for reinjection (see keywords GINJGAS, WINJGAS, GRUPINJE and WELLINJE). When gas from this well is reinjected, the advance gas is used for reinjection prior to the gas produced by this well from the reservoir. If the reinjection requirements can be met by the advance gas stream alone, then no produced gas is reinjected. If the advance and available produced gas streams together are not sufficient to satisfy the reinjection requirements, make-up gas, if specified for the injection group or well, are used to make up the shortfall.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/). The set of records must be ended by a blank record, containing only a slash.

1. Well name, well name root or well list name
   A well name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several wells in one record. Alternatively a well list name, enclosed in quotes and beginning with an asterisk (*), may be used. Well lists are constructed with the keyword WLIST.

2. The name of the wellstream (defined using the WELLSTRE keyword) which defines the composition of the advance gas.

3. The maximum rate at which advance gas can be supplied.
   UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)
   By setting the item 30 in the OPTIONS3 keyword, an alternative mixing tank treatment may be invoked, where the advance gas is mixed with the produced gas and gas from this mixing tank is reinjected. The fraction of advance gas in this mixture is equal to the ratio of the specified advance gas rate (item 3) to the sum of the specified advance gas rate and the available gas production rate. With this option, the advance gas is not used in preference to the produced gas.

   If advance gas is used, the injection calculation may have to be iterated to match the injection gas composition with the injection rate (see keyword GCONTOL).

Note: Advance gas is assigned to the source well supplying the gas, while make-up gas is assigned to the group or well having the injection target.

Example

Advance gas is assigned to well W3 at a maximum rate of 4100 Mscf/day.

WADVANCE
W3 STR1 4100 /
WAGHALT  Select the alternative WAG hysteresis model

This indicates that the alternative WAG hysteresis model is to be used. This corresponds more closely with the ECLIPSE 100 WAG hysteresis model as compared to the default ECLIPSE 300 WAG hysteresis model. This keyword requires no data items and should be terminated with a slash (/). The hysteresis models are described in more detail under "Hysteresis in WAG floods" in the ECLIPSE Technical Description.

Example

Alternative WAG hysteresis model example

```
WAGHALT
/
```
WAGHYSTR  WAG hysteresis parameters

This keyword activates the WAG hysteresis model and supplies the data items required by the model. The WAG hysteresis model can only be used if the hysteresis option has been requested (see keyword SATOPTS in the RUNSPEC section).

The keyword should be followed by NTSFUN (see keyword TABDIMS in the RUNSPEC section) records of data, each terminated by a slash (/). The entire table may be defaulted provided the table is not the first one. Defaulted tables are replaced with a copy of the previous table. The NTSFUN records apply to the corresponding saturation table numbers.

Each record should consist of some or all of the following items of data:

1. Land’s parameter, C
   This parameter governs the trapped gas saturation on imbibition and the shape of the imbibition curve. The trapped gas saturation is given by:
   \[
   S_{g_{trap}} = S_{g_{cr}} + \frac{(S_{g_{m}} - S_{g_{cr}})}{(1 + C \cdot (S_{g_{m}} - S_{g_{cr}}))}
   \]
   where
   - \( S_{g_{trap}} \) is the trapped gas saturation
   - \( S_{g_{m}} \) is the maximum gas saturation attained
   - \( S_{g_{cr}} \) is the critical gas saturation.
   Land’s parameter should be greater than zero. If the parameter is too small the trapped gas saturation is close to \( S_{g_{m}} \) and the subsequent imbibition relative permeability curve is steep. An unphysically steep curve is likely to lead to convergence problems.

2. The Secondary Drainage reduction factor, \( \alpha \)
   The reduction factor should be greater than zero. The greater the value of \( \alpha \) the greater will be the reduction in gas mobility on secondary drainage.
   DEFAULT: 0.0

3. Gas model flag
   YES  Use the WAG Hysteresis Model for the gas phase relative permeability.
   NO   Turn off the WAG Model and use the drainage curve.

   DEFAULT: YES

4. Residual oil flag
   YES  Use the trapped gas saturation to modify the residual oil (SOM) in the STONE 1 three-phase oil relative permeability model. No action is taken unless the STONE1 keyword has been entered.
   NO   Do not modify the oil relative permeability.

   DEFAULT: YES

Keywords

ECLIPSE Reference Manual

2416
5. Water model flag

YES  Apply the WAG Hysteresis Wetting Phase Model to the water phase.

NO   Do not apply the WAG hysteresis model. Note that the Hysteresis Model specified in the EHYSTR keyword applies.

DEFAULT: YES

6. Imbibition curve linear fraction

This is the fraction of the curve between $S_{gm}$ and $S_{gtrap}$ that uses a linear transformation. A non-zero value for the linear fraction prevents the potential infinite gradient in the imbibition curve when using the Carlson analytic model.

DEFAULT: 0.1

7. Three-phase model threshold saturation

This is the water saturation threshold above the connate value at which the gas (non-wetting) phase hysteresis switches from the two-phase model to the three-phase model. In the two-phase model a secondary drainage process follows the imbibition curve. However, if the water saturation exceeds the connate saturation by the given threshold, at the beginning of the secondary drainage process a three-phase secondary drainage curve is followed. This value is also used as a percentage change in gas saturation to allow switching from drainage to imbibition curve and vice-versa. This threshold allows better control of the numerical sensitivity of the system, preventing it from being too unstable.

DEFAULT: 0.001

8. Residual oil modification fraction

This is the fraction of the trapped gas saturation subtracted from the residual oil ($SOM$) in the STONE 1 three-phase oil relative permeability model. The value should lie between zero and 1.

DEFAULT: 1.0

The hysteresis models are described in more detail under "Hysteresis in WAG floods" in the ECLIPSE Technical Description.

Notes

- In some situations the switching from the drainage to imbibition curve and vice-versa can be so sensitive to numerical errors that the loss of precision at a restart is enough to give differing answers between the base case and the restart run. In these exceptional cases it is recommended not to use the restart simulations at the points of switching curves. This allows better control of the numerical sensitivity of the system, preventing it from being different to the base simulations.

- The usage of the gas and water relative permeability versus saturation tables differs between conventional hysteresis and WAG hysteresis. For conventional hysteresis, the imbibition tables define the imbibition saturation functions. For WAG hysteresis, if this is applied to the gas phase, the imbibition table gas phase relative permeability specifies the three-phase relative permeability rather than the imbibition relative permeability. Similarly, if this is applied to the water phase, the imbibition table water relative permeability specifies the three-phase relative permeability. Hence the process of selecting or deselecting WAG hysteresis will, in general, require changes to the imbibition saturation function tables for the gas and water relative permeabilities.
ECLIPSE 100

- WAGHYSTR cannot be specified in a restart run unless it has been specified in the base run and any intermediate restart runs.

ECLIPSE 300

- Versions prior to 2012.1 did not support parallel restarts if the WAGHYSTR keyword was specified. For the 2012.1 version, the restart records associated with the WAG hysteresis option have been restructured in order to support the various possible combinations of serial/parallel base case and serial/parallel restart case and similarly for successive restarts.

- For the 2012.1 version onwards, WAGHYSTR can be specified in a restart run without having been specified in the base run and vice versa. A similar relaxation applies to successive restarts. However, careful consideration should be given as to whether a temporally partial modeling of WAG hysteresis is appropriate for a given reservoir model. In particular, if WAGHYSTR is first specified in a restart run, no previous WAG history will be available and the simulation results will be different to those where WAG hysteresis has been modeled throughout. Similarly, if WAGHYSTR is specified in the base case but not in the restart case, the WAG history from the base case will not be preserved or used for the restart run and the simulation results will again be different to those where WAG hysteresis has been modeled throughout.

- For versions prior to 2012.1, the WAG hysteresis restart record written by a parallel base case has been found in some cases to be incomplete and problems of this kind are now detected and reported. In such cases, the recommended workaround is to rerun the base case in serial using the previous version or in serial or parallel using the 2012.1 or later version. Alternatively, the incomplete WAG hysteresis restart record may be bypassed by setting item 279 of the OPTIONS3 keyword to 2. However, in this case, the previous WAG history will be lost.

- When reading from a pre-2012.1 restart record, it is not possible to perform a parallel restart run. Only a serial run is possible. This limitation is inherited from the restriction on pre-2012.1 version WAG hysteresis records for which parallel restarts are not supported. If a parallel restart from a pre-2012.1 version is required, the recommended workaround is to perform an initial short serial run and write a 2012.1 version restart record. This may then be used to run any subsequent restarts in parallel.

- The use of the restructured WAG hysteresis records for the 2012.1 version can be reverted to the pre-2012.1 restart record type by setting item 279 of the OPTIONS3 keyword to 1. However, in this case, all of the limitations which apply to the pre-2012.1 versions will also be reinstated, in particular, no support for parallel restarts, potentially incomplete parallel base case restart records and no facility for selecting or deselecting WAG hysteresis with each successive restart.

- In some cases it has been found that reading restarts which contain WAG hysteresis records created by the 2009.1 and 2009.2 versions fails with a message referring to an incorrect header record. This is associated with an attempt to read a UDA/UDQ restart record when none was defined. Under these circumstances item 282 of the OPTIONS3 keyword may be set to 1 to bypass the reading of the nonexistent UDA/UDQ record thereby resulting in the correct reading of the WAG hysteresis record. (This does not address the problems associated with the incorrect size of the WAG hysteresis which can occur in some cases with restarts written by parallel runs using versions prior to 2012.1.)
Example

With NTSFUN=3:

WAGHYSR
2.0 1.0 YES YES YES 0.2 /
2.0 1.0 YES NO /
2.0 /
**WAITBAL**

Prevents PRORDER and GDRILPOT actions being performed until network is balanced

The keyword can be used in conjunction with the network option (see "Network option" in the ECLIPSE Technical Description) to request that the drilling of new wells and other production rule actions to enhance production should not occur until the network has been balanced.

At each network balancing iteration, the well and group production rates are determined at the latest iterate of the nodal pressures. If a group production target cannot be met, actions may be taken to increase production, such as drilling a new well (see keyword PRORDER). New wells may also be drilled to maintain a group’s production potential above a specified value (see keyword GDRILPOT). By default, all such actions except for automatic compressor switching (keyword NETCOMPA) are performed whenever required to increase the production rate in any network balancing iteration. However, it is possible that an intermediate balancing iteration with high nodal pressures and consequentially low well productivities may cause such an action to be taken prematurely. This can be prevented by using the WAITBAL keyword to request that these actions should not be performed until the network is properly balanced. Then if more flow is required once the network has converged, an appropriate action is performed and the network must be balanced again. Using this option therefore requires more network balancing iterations in total.

The keyword is followed by one item of data, terminated with a slash (/).

1. Flag indicating whether PRORDER and GDRILPOT actions should wait until the network is balanced.
   
   **NO**  The actions are performed at any iteration of the network balancing calculation.
   
   **YES**  The actions are only performed if required after the network balancing iterations have converged.

   (The word may be abbreviated; only the first character is significant)

   **DEFAULT:** NO

   End the data record with a slash (/).

   **Note:** ECLIPSE 300 always waits until the network is balanced before deciding whether to open a new well from the drilling queue; this is equivalent to having item 1 permanently set to YES.

**Example**

```
WAITBAL
YES /
```
WALKALIN  Sets alkaline concentration for water injection wells

This keyword is used to specify the concentration of alkaline in the water injection well when the surfactant flood model and/or the polymer flood model are active.

The keyword is followed by any number of records, each containing the following data:

1. Well name, well name template, well list or well list template.

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. The concentration of alkaline in the water injection stream. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

   UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scce (LAB), kg/sm$^3$ (PVT-M)

End data record with a slash (/).

End the set of records with a blank record, containing just a slash (/).

Example

```
WALKALIN
 INJ1  0.3 /
 INJ2  0.2 /
```
WALQCALC  Defines the well ALQ to be the oil or gas surface density

This keyword can be used to equate the ALQ value in a well to its oil surface density or its gas surface density. VFP i can produce a VFP table that spans a range of oil or gas surface densities, and the WALQCALC keyword instructs ECLIPSE to interpolate it at the current surface density of the oil or gas that the well is producing, by setting its ALQ equal to that value. This option is useful in API tracking runs, and also when the same table is used by several wells in different PVT regions. (A similar option can also be applied to branch pressure drops in the network option; see “Network option” in the ECLIPSE Technical Description, and see also keywords GRUPNET and BRANPROP.)

The WALQCALC keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLST.

   A well name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several wells in one record. Alternatively a well list name, enclosed in quotes and beginning with an asterisk (*), may be used. Well lists are constructed with the keyword WLST.

2. ALQ definition for the well

   A well name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several wells in one record. Alternatively a well list name, enclosed in quotes and beginning with an asterisk (*), may be used. Well lists are constructed with the keyword WLST.

   DENO  The well’s ALQ is equated to its oil surface density, in kg/m^3 (METRIC), lb/ft^3 (FIELD), gm/cc (LAB), kg/m^3 (PVT-M)

   DENG  The well’s ALQ is equated to its gas surface density, in kg/m^3 (METRIC), lb/ft^3 (FIELD), gm/cc (LAB), kg/m^3 (PVT-M)

   NONE  The well’s ALQ will not be automatically set equal to its oil or gas surface density

   DEFAULT: NONE

End the data record with a slash (/).

End the set of records with a blank record containing just a slash.

Note: If DENO or DENG is chosen for a well, the VFP table that it uses must be constructed using the same definition of the ALQ in item 7 of the first record of the VFPPROD keyword.
CAUTION: Obviously the ALQ cannot also be used for artificial lift when it is equated to the well’s oil or gas surface density. Thus the surface density definition must not be used in circumstances when the ALQ is assumed to be the lift gas injection rate (for example in Gas Lift Optimization, or with GLIFTLM).

Example

```
WALQCALC: "*" DENO       / equates ALQ to oil surface density in all wells
       PROD3  NONE       / except for PROD3 which has no special ALQ definition
```

Keywords

2423
WAPI

Sets oil API for oil injection wells

This keyword can be used to specify the oil API in the injection stream of each well. The keyword should only be used in runs with the API tracking option active (keyword API in the RUNSPEC section). If the WAPI keyword does not appear in the injection schedule, then the API of the block being injected into is assumed. Data should only appear for wells which are currently declared to be oil injection wells in WCONINJE.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. The corresponding value of the injected oil API gravity. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

   UNITS: °API

End data record with a slash (/).

End the set of records with a blank record, containing just a slash (/).

Example

```
WAPI
OILINJ  42.0 /
/
```
WARN

Allow ECLIPSE warning messages

This keyword switches warning messages back on after being suppressed by the `NOWARN` keyword.

The keyword has no associated data.
Enable WARP linear solver

Before the 2002A release, the default solver was ECLSOL, and this keyword was used to switch to the WARP solver. From the 2002A release onwards, the WARP solver is the default, so this keyword now has no effect. To switch to the ECLSOL solver use item 66 of the OPTIONS3 keyword.

See "The ECLIPSE 300 WARP linear solver" in the ECLIPSE Technical Description for more details.
**WATDENT**  

**Water density as a function of temperature**

This keyword is optional, and may only be used in runs with the Thermal option. See keyword THERMAL.

The data comprises NTPVT (see keyword TABDIMS) records, each terminated by a slash (/). Each record consists of three items of data:

1. The reference temperature $T_{ref}$
   
   **UNITS:** K (METRIC), °R (FIELD), K (LAB), K (PVT-M)
   
   **DEFAULT:** 293.15 K = 527.67 °R

2. The first thermal expansion coefficient $c_{T1}$
   
   **UNITS:** 1/K (METRIC), 1°R (FIELD), 1/K (LAB), 1/K (PVT-M)
   
   **DEFAULT:** 3.0e-4 K$^{-1}$ = 1.67 °R$^{-1}$

3. The second thermal expansion coefficient $c_{T2}$
   
   **UNITS:** 1/K$^2$ (METRIC), 1/°R$^2$ (FIELD), 1/K$^2$ (LAB), 1/K$^2$ (PVT-M)
   
   **DEFAULT:** 3.0e-6 K$^{-2}$ = 9.26e-7 °R$^{-2}$

If this keyword is used the water formation volume factor $B_w$ is defined by

$$B_w (P, T) = B_w (P_{ref}) \cdot (1-X) \cdot (1 + c_{T1} \cdot (T-T_{ref}) + c_{T2} \cdot (T-T_{ref})^2)$$

where

$$X = C \cdot (P-P_{ref})$$

$P_{ref}$, $B_w (P_{ref})$ and $C$ are defined by the PVTW keyword.

**Example**

```
WATDENT
```

---

**Keywords**

2427
Run contains water

This indicates that the run contains water as an active phase, whose saturation can vary. The keyword has no associated data.
WATERTAB  Water pressure tables

The data specified consists of a number of tables of water pressure data, each terminated with a slash (/). The number of tables should be equal to NTPVT (see item 2 of the TABDIMS keyword).

Each row of the table consists of 3 columns of data, and at least two rows of data should be entered.

Outside the range of specified pressures, values are linearly extrapolated.

1. The water phase pressure.
   
   Values should increase down the column.
   
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2. The water formation volume factor

   Values should decrease down the column.

   UNITS: rm³/sm³ (METRIC), rb/stb (FIELD), rcc/scc (LAB), rm³/sm³ (PVT-M)

3. The water viscosity.

   Values should be level or increasing down the column.

   UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

There must be the same number of entries in each column of a given table, and the number of rows should be less than or equal to NPPVT (item 4 of keyword TABDIMS).

Apart from the last or first rows, values in the second and third columns may be defaulted, and such values are linearly interpolated.

Example

<table>
<thead>
<tr>
<th>Pressure</th>
<th>Water Phase Pressure</th>
<th>Formation Volume Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>3000</td>
<td>1.00341</td>
<td>0.52341</td>
</tr>
<tr>
<td>3900</td>
<td>1</td>
<td>0.56341 /</td>
</tr>
</tbody>
</table>
**WATVISCT**  
**Water viscosity as a function of temperature**

This keyword is optional and can be used with the blackoil temperature option or the compositional Thermal option.

If the temperature option is active (see keyword TEMP), and if the WATVISCT keyword is used, then the VISCREF keyword should be specified to supply the reference conditions for the water viscosity. If the Thermal option is active (see keyword THERMAL), VISCREF is only needed if a pressure dependence is required, as discussed below.

The data comprises NTPVT (see keyword TABDIMS) tables of water viscosity versus temperature, each terminated by a slash (/). Each table consists of 2 columns of data:

**Column:**

1. **The temperature.**
   - The values should increase monotonically down the column.
   - **UNITS:** °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

2. **The corresponding water viscosity**
   - **UNITS:** cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

There must be the same number of entries in each column of a given table. This number should not be less than 2 or greater than NPPVT (see keyword TABDIMS).

The values supplied for the water viscosity are assumed to be at the reference pressure given in the VISCREF keyword and at the reference salt concentration defined by item 2, record 1 of the PVTWSALT keyword, if the Brine option is active. The water viscosity at the prevailing temperature, pressure and optionally salt concentration is then calculated as follows:

\[
\mu_w(T, P) = \mu_T(T) \frac{\mu_p(P)}{\mu_p(P_{ref})}
\]

Eq. 3.213

or, if the brine option is active, as

\[
\mu_w(T, P, c_s) = \mu_T(T) \frac{\mu_p(P, c_s)}{\mu_p(P_{ref}, c_{s\, ref})}
\]

Eq. 3.214

where

- \( \mu_T \) is the viscosity from the WATVISCT keyword.
- \( \mu_p \) is the viscosity from the PVTW or PVTWSALT keyword.
- \( P_{ref} \) is the reference pressure defined by the VISCREF keyword.
- \( c_{s\, ref} \) is the reference salt concentration defined by item 2, record 1 of the PVTWSALT keyword.

In the Thermal VISCREF the values of water viscosity are assumed to be independent of pressure, unless the VISCREF keyword is used. In this case the water viscosity is defined by equation 3.213.

See also keyword OILVISCT.
ECLIPSE 100

**Note:** ECLIPSE 100 linearly interpolates the reciprocal of the water viscosities between data points, rather than the values themselves.

---

ECLIPSE 300

**Note:** ECLIPSE 300 linearly interpolates the water viscosities directly between data points.

---

**Example**

With NTPVT =1 and NPPVT ≥3:

```plaintext
WATVISCT
   10  1
   100 0.6
   150 0.4
```

---

Keywords

2431
**WAVAILIM**

**Requests availability limiting of injection**

This keyword requests that the availability of injection fluids is to be taken into account when calculating injection rates. If the required injection target is greater than the amount of injection fluid available from the production system the injector goes onto availability control.

If the injection stream originates from a particular source group or well (named in item 3 of GINJGAS, WINJGAS, GRUPINJE or WELLINJE when item 2 is GV or WV), and the injection target exceeds the available gas from this source, the extra gas required is taken from the make-up gas stream if this is specified for the group or well with the injection target. If no make-up gas is specified, the behavior of the injection controls will depend on whether the keyword WAVAILIM is entered. Without this keyword, the injection target is honored by injecting more gas than is available, resulting in a negative excess rate in the gas accounting table. If the WAVAILIM keyword is entered, the injection rate is limited by the amount of gas available for injection from the source group or well. When the injection stream originates from a source group, the gas available for injection is its produced gas, minus its fuel and sales gas (unless specified otherwise in WTAKEGAS), plus its advance gas rate specified in GADVANCE (unless the 99A release default mixing tank treatment has been enabled by setting item 30 in OPTIONS3 to 2). When the injection stream originates from a source well, the gas available for injection is its produced gas, plus its advance gas specified in WADVANCE (unless the 99A release default mixing tank treatment has been enabled by setting item 30 in OPTIONS3 to 2).

If a limited source of gas is shared between two or more injection groups, the injection calculation may have to be iterated a number of times to match the groups’ share of the available gas with their injection capability (see keyword GCONTROL).

The WAVAILIM keyword contain no items.

See also "Group injection control options" in the ECLIPSE Technical Description.
Well limits on bottom hole gas liquid ratio

This keyword provides two facilities for limiting the bottom hole gas liquid ratio of a production well. The bottom hole GLR is defined as the ratio of free gas flow to liquid flow at bottom hole conditions, that is at the bottom hole pressure of the well (at the datum defined in item 5 of keyword WELSPECS). Facilities to restrict this quantity can be useful when there is a down hole pump, whose efficiency can be seriously reduced by the presence of free gas. Using this keyword, the quantity can be restricted by a well cutback limit and/or a well economic limit.

The well cutback limit is applied in exactly the same manner as the WCUTBACK keyword. If the well exceeds the cutback limit (item 2) at the end of the timestep, it will be set to operate at a reduced target rate of the control phase specified in item 5. The target rate of the control phase is set equal to the well’s current flow rate of that phase, multiplied by the cutback factor specified in item 4. A cutback reversal limit may also be entered (item 3) to reverse the cutback process if the bottom hole GLR falls below this value. If the well is currently operating under rate control of the phase selected in item 5, the target rate of the phase is increased by dividing it by the cutback factor. Any user-defined limit on the control phase in item 5 is overwritten, so it is best to choose as the control phase a phase for which there are no user-defined limits.

The economic limit (item 6) is applied in exactly the same manner as the WECON keyword. If the bottom hole GLR exceeds the economic limit, the well is closed or worked over as specified in item 7.

The value of the bottom hole GLR can be written to the Summary file using the SUMMARY section mnemonic WBGLR.

The WBHGLR keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Upper limit bottom hole GLR for well rate cutback
   A value of 0.0 or > 1.0E19 switches off this check.
   UNITS: rm³/rm³ (METRIC), rb/rb (FIELD), rcc/rcc (LAB), rm³/rm³ (PVT-M)
   DEFAULT: Infinity (1.0E20)

3. Lower limit bottom hole GLR for reversing the cutback process
   A value of 0.0 or > 1.0E19 switches off this check.
   UNITS: rm³/rm³ (METRIC), rb/rb (FIELD), rcc/rcc (LAB), rm³/rm³ (PVT-M)
   DEFAULT: 0.0

4. Rate cutback factor
   This is the ratio of the reduced flow rate to the current flow rate. It should be less than 1.0
   The same rate cutback factor applies to both this keyword and the WCUTBACK keyword. If WCUTBACK has been entered previously, this item overwrites item 6 in that keyword. If WCUTBACK is entered subsequently, its item 6 overwrites this item.
5. Control phase, to which the reduced rate applies
   - OIL  Reduced oil rate
   - WAT  Reduced water rate
   - GAS  Reduced gas rate
   - LIQ  Reduced liquid rate
   - RESV Reduced reservoir fluid volume rate
   - COMB Reduced linearly combined rate (coefficients must be set with LINCOM)

You are advised to choose a phase that does not have a user defined rate limit, as it would be overwritten whenever a cutback or cutback reversal occurs; see Note 3 in the documentation for keyword WCUTBACK.

The same control phase applies to both this keyword and the WCUTBACK keyword. If WCUTBACK has been entered previously, this item overwrites item 7 in that keyword. If WCUTBACK is entered subsequently, its item 7 overwrites this item.

DEFAULT: same as item 7 in the WCUTBACK keyword, if that has been entered previously, otherwise RESV

6. Maximum bottom hole GLR economic limit
   A value of 0.0 or > 1.0E19 switches off this check.
   UNITS: rm³/rm³ (METRIC), rb/rb (FIELD), rcc/rcc (LAB), rm³/rm³ (PVT-M)
   DEFAULT: Infinity (1.0E20)

7. Workover procedure on exceeding the bottom hole GLR economic limit
   - NONE  Do nothing
   - CON   Shut worst-offending connection
   - +CON  Shut worst-offending connection and all below it
   - WELL  Shut or stop well (item 9 in keyword WELSPECS)
   - PLUG  Plug back well (see keyword WPLUG)

DEFAULT: same as item 7 in the WECON keyword, if this has been entered previously; otherwise NONE

8. Remove cutbacks when the well is worked over?
   - YES  When the well is worked over after violating an economic limit (set in keywords CECON and WECON for example), any rate cutbacks performed on the well are completely reversed by removing the rate limit on the control phase specified in item 5.
Any cutbacks made to the well remains in force (unless reversed by application of cutback reversal limits).

The same instruction applies to both this keyword and the WCUTBACK keyword. If WCUTBACK has been entered previously, this item overwrites item 14 in that keyword. If WCUTBACK is entered subsequently, its item 14 overwrites this item.

DEFAULT: NO

End each data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also keywords WCUTBACK, WECON and WLIMTOL.

Example

| WBHGLR | | | | | | |
| Well | Cutback | Cutback | Cutback | Control | Economic | Workover |
| Name | Limit | Reversal | Factor | Phase | Limit | Procedure |
| WELL1 | 0.3 | 0.05 | 0.8 | RESV | / | / |
| WELL2 | 4* | | | | 0.5 | CON | / |

Keywords

2435
WBOREVOL  Sets volume for wellbore storage

This keyword specifies an effective wellbore volume for individual wells, to enable ECLIPSE to model wellbore storage effects in well tests. ECLIPSE calculates the well solution variables (BHP, water fraction and gas fraction) by solving a material balance equation for each phase in each well:

production rate = inflow through connections - material accumulation in wellbore.

Normally, the accumulation term is made negligible by giving the wellbore a very small volume (1.0E-5 FT^3). But the keyword WBOREVOL allows you to specify a larger volume, enabling ECLIPSE to model wellbore storage effects through the accumulation term. The keyword should be entered before the WCONPROD or WELOPEN keyword that opens or stops the well at the start of the test. Note that in order to calculate the BHP variation in a shut-in test, the well must be stopped rather than shut.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Effective wellbore volume

   UNITS: m^3 (METRIC), ft^3 (FIELD), cc (LAB), m^3 (PVT-M)

   DEFAULT: 1.0E-5, giving negligible storage effects.

3. Well BHP at start of test

   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

   DEFAULT: Pressure in the grid block containing the topmost connection in the well. (This is a suitable value for the start of the flowing period, provided that the bottom hole datum depth has been defaulted in item 5 of keyword WELSPECS. If the bottom hole datum is set at any other depth, you should either set the initial BHP here if it is known, or let the well reach its initial BHP by running it stopped for a while before the test.)

   The setting of the initial BHP is ignored if the wellbore volume is 1.0E-5 or less, as this turns the wellbore storage facility off.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Note: This keyword must not be used for multisegment wells (see "Multisegment wells" in the ECLIPSE Technical Description). Their wellbore volumes are set when their segments are defined in keyword WELSEGS.

Example

```
WBOREVOL
   -- well name  wellbore volume  initial pressure
```

Keywords
The effective wellbore volume

The effective wellbore volume can be determined from the dimensionless wellbore storage coefficient, as shown below.

The dimensionless wellbore storage coefficient, $C_D$, is defined as:

$$C_D = \frac{C}{2\pi \varphi c_t h r_w^2}$$  \hspace{1cm} \text{Eq. 3.215}$$

where

$C$ is the wellbore storage coefficient

$\pi$ is 3.14159

$\varphi$ is the formation porosity

$c_t$ is the total compressibility of the system (fluid + rock)

$h$ is the formation thickness

$r_w$ is the wellbore radius.

The wellbore storage coefficient $C$ is defined as the increase in the volume of fluid that the wellbore can store per unit increase in pressure,

$$V = \frac{dV}{dP}$$  \hspace{1cm} \text{Eq. 3.216}$$

For storage due to fluid expansion in the wellbore,

$$C = c_w V_w$$  \hspace{1cm} \text{Eq. 3.217}$$

where

$c_w$ is the compressibility of the fluid in the wellbore

$V_w$ is the effective volume of the wellbore.

Thus the effective wellbore volume is related to the dimensionless storage coefficient by:

$$V_w = \frac{2\pi \varphi c_t h r_w^2 C_D}{c_w}$$  \hspace{1cm} \text{Eq. 3.218}$$
WCALVAL  Sets the calorific value of gas at a well

The keyword WCALVAL specifies the calorific value of the gas produced from a particular well, for use with the gas calorific value control option (see “Gas calorific value control” in the ECLIPSE Technical Description). The energy production rate from that well is then equal to its gas rate multiplied by its calorific value. Well calorific values can be changed at any time by reentering the keyword.

If no tracer has been defined with the CALTRAC keyword to represent the calorific value of the gas in the reservoir, the gas producing wells must have their calorific values set individually with keyword WCALVAL. Any well not given a calorific value with keyword WCALVAL will have a zero calorific value.

If a calorific value tracer has been defined with keyword CALTRAC, each well’s calorific value will be set equal to its produced concentration of the tracer, unless this is overridden by use of the WCALVAL keyword. WCALVAL may be used to override the tracer value for individual wells. A well may subsequently be switched back to using the tracer value by entering a negative value for the calorific value in item 2.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Calorific value of gas from the well
   A negative value causes the well to have its calorific value set according to the concentration of the tracer named in the CALTRAC keyword.
   
   UNITS: kJ/sm$^3$ (METRIC), Btu/Mscf (FIELD), J/sec (LAB), kJ/sm$^3$ (PVT-M)
   DEFAULT: -1.0

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Example

```
WCALVAL
  'P*'      40000.0  /
  PROD3     38500.0  /
  PROD4     39300.0  /
```

Keywords

2438
Coloring for WARP linear solver

This keyword controls the coloring method used within the WARP linear solver. This is initiated using the WARP keyword. Coloring is required when using the PARALLEL option. The presence of this keyword initiates the WARP linear solver. The keyword contains three items of data terminated with a slash (/).

1. The type of coloring.
   Valid inputs are:
   - 'XY' The partition is subdivided in the X and Y directions
   - 'X' The partition is subdivided in the X direction
   - 'Y' The partition is subdivided in the Y direction
   - If left blank the coloring is chosen based on the partition splits, so if NPROCX, NPROCY then ‘Y’ coloring is used, otherwise ‘X’ coloring is used. This is the DEFAULT.

2. Number of colors.
   This item is only relevant if the number of processors is set greater than 1 in item 1 of the PARALLEL keyword. Otherwise, a value of 1 is assumed.
   DEFAULT: 2 (if there is more than one processor)

3. LGR sub-grid treatment.
   1 - The color of any LGR grid is inherited from the base grid. Portions of LGR grid of the same level of refinement and color are amalgamated into a sub-grid.
   If the number of colors set in item 2 of this keyword is set to 1, then each level of LGR refinement represents a sub-grid.

   2 - The color of any LGR grid is inherited from the base grid, and the portions of the base grid and LGR grid that have the same color are then amalgamated into the same sub-grid. This is the DEFAULT.
   In this case the number of sub-grids is independent of the number, or levels, of LGRs. If the number of colors in item 2 of this keyword is set to 1, then only one sub-grid will result.

   3 - Each LGR is colored using the type and number specified in items 1 and 2. No amalgamation of LGRs within or between levels is performed.
   Many sub-grids can easily be generated with this method.

For further information see "The ECLIPSE 300 WARP linear solver" in the ECLIPSE Technical Description

Example

```
WCOL
  'Y' 3 /
```
WCOMPRES  Compresses internal well arrays

This keyword may be used to reduce to memory requirements of a simulation with a large number of wells, some of which have many connections, but most have relatively few connections. In such circumstances, internal well arrays holding connection data that are dimensioned to the maximum number of connections per well multiplied by the maximum number of wells (see items 1 and 2 of WELLDIMS) will be wasteful of memory. If this keyword is specified, these internal arrays will be dimensioned to the actual number of well connections in the simulation, which may be specified in item 1 below. Alternatively, if item 1 is defaulted, a pre-sweep of the SCHEDULE section keywords will be performed to estimate the number of well connections in the simulation. Both the actual and estimated number of connections are reported in the print file at the end of the simulation.

The keyword is followed by one record containing one item and must be terminated with a slash (/).

1. Number of well connections in the simulation.

   DEFAULT: This quantity will be estimated by a pre-sweep of the Schedule section keywords that define wells and well connections.

   **Note:** In some circumstances, the number of connections estimated by the pre-sweep or set by the user may be less than the actual number of connections required during the simulation. For example, this will happen if new connections are defined in a flexible restart run that are not present in the base run. (A pre-sweep of the restart data may not be reliable, and is thus banned, as some or all of the SCHEDULE section keywords may have been deleted before the timestepping keyword at the restart time.) In these circumstances, extra space for the internal well arrays will be automatically reallocated by increasing the number of connections in blocks of ten.

**Example**

```
WCOMPRES /
```
WCONHIST  Observed rates for history matching production wells

This keyword is used in place of WCONPROD to declare production wells as special history matching wells, and to enter their observed flow rates (and optionally their measured BHP and THP values). The equivalent keyword for defining history matching injection wells is WCONINJH.

Wells can be declared as history matching wells during the history matching process, when their oil, water and gas production rates are known. ECLIPSE treats them differently from ordinary production wells, in the following ways:

- The observed oil, water and gas rates entered with this keyword can be written to the summary file, along with the water cut, GOR and THP for example, for comparison with the rates calculated during the simulation.

- The wells have a restricted set of control modes:
  
  ORAT  The well produces the observed oil rate. The other phases will be produced according to their mobility ratios.
  
  WRAT  The well produces the observed water rate. The other phases will be produced according to their mobility ratios.
  
  GRAT  The well produces the observed gas rate. The other phases will be produced according to their mobility ratios.
  
  LRAT  The well produces at a liquid rate equal to the sum of the observed oil and water rates. The calculated oil and water rates may differ from the values entered if the mobility ratios have not been fully matched.
  
  CRAT  The well produces at a combined rate equal to the linear combination of the observed oil, water and gas rates. The coefficients for the linear combination are defined with keyword LINCOM.
  
  RESV At each timestep, ECLIPSE calculates the reservoir fluid volume rate corresponding to the observed phase flow rates, using the reservoir pressure at the beginning of the timestep, and puts the well under RESV control to produce at that target. (The reservoir pressure used is either the average hydrocarbon pressure of the field, or the average hydrocarbon pressure of a specified fluids-in-place region, depending on the entry for item 13 in the keyword WELSPECS.) The calculated oil, water and gas rates may differ from the values entered if the mobility ratios have not been fully matched.
  
  WGRA The well produces the observed wet gas rate. The other phases will be produced according to their mobility ratios.
  
  NGL The well produces the observed NGL rate. The other phases will be produced according to their mobility ratios.
  
  BHP The well produces at the observed BHP.

The LRAT and RESV control modes are useful for making the well produce the correct amount of total fluid from the reservoir before the mobility ratios are fully matched. Thus the rate of pressure decline should be approximately correct.
• The phase flow rates entered here are not treated as upper limits. If the well’s calculated water rate, for example, exceeds the observed value, the well does not automatically change control to WRAT. Apart from setting the rate target, the phase flow rates are only used for reporting in the SUMMARY file.

• For control modes other than BHP the BHP lower limit is automatically set to atmospheric pressure when the well is first declared a history matching well (the first time it appears in a WCONHIST keyword). This is to reduce the possibility of the well changing to BHP control before its productivity index has been properly matched. (Changing to BHP control at atmospheric pressure is a sign that the Kh or skin factor is very badly matched.) You can reset the BHP limit to any desired value with the keyword WELTARG after the first WCONHIST keyword in which the well appears. The well’s BHP limit is not changed by subsequent WCONHIST keywords unless the BHP control mode is selected. There is an option (see keyword WHISTCTL) to make the run stop automatically if a history matching well changes to BHP control.

**CAUTION:**

With multisegment wells (keyword WELSEGS), you are recommended to use WELTARG after WCONHIST to set a BHP limit that is within the range of the PVT tables. If these tables are extrapolated to atmospheric pressure, it is quite possible that the fluid properties could become unphysical. The multisegment well model requires realistic fluid properties in order to converge. Note that the calculation of the well potential uses the BHP limit, even if the actual operating BHP of the well is far above this value.

• For BHP control mode the observed BHP is set as the target. If a subsequent WCONHIST keyword alters the control mode then the BHP target will be reset to atmospheric pressure.

The observed flow rates and pressures can be made to change with time by repeating the WCONHIST keyword at each timestep. A history matching well can be converted into a normal well (with the standard set of controls) simply by re-specifying it with the WCONPROD keyword.

The WCONHIST keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash.

1. Well name, well name template, well list or well list template
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Open/shut flag for the well
   
   OPEN Well open for production
   
   STOP Well stopped off above the formation
   
   SHUT Well completely isolated from the formation

   DEFAULT: OPEN

   When the well is shut or stopped, set the observed flow rates to zero. If the flow rates are set to zero while the well is open the behavior depends on whether crossflow is permitted (item 10 in keyword WELSPECS). If crossflow is allowed the well behaves as if stopped (although its status is still reported...
as OPEN). If crossflow is banned the well will be automatically shut (and its status is reported as SHUT).

3. Control mode

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ORAT</td>
<td>Controlled by the observed oil rate (item 4)</td>
</tr>
<tr>
<td>WRAT</td>
<td>Controlled by the observed water rate (item 5)</td>
</tr>
<tr>
<td>GRAT</td>
<td>Controlled by the observed gas rate (item 6)</td>
</tr>
<tr>
<td>LRAT</td>
<td>Controlled by the observed liquid rate (oil + water)</td>
</tr>
<tr>
<td>CRAT</td>
<td>Controlled by the observed linearly combined rate (see keyword LINCOM)</td>
</tr>
<tr>
<td>RESV</td>
<td>Controlled by the reservoir fluid volume rate calculated from the observed phase flow rates</td>
</tr>
<tr>
<td>BHP</td>
<td>Controlled by the well bottom hole pressure (item 10)</td>
</tr>
</tbody>
</table>

**ECLIPSE 300 only**

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>WGRA</td>
<td>Controlled by the observed wet gas rate (item 11)</td>
</tr>
</tbody>
</table>

**ECLIPSE 300 only**

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NGL</td>
<td>Controlled by the observed NGL rate (item 12)</td>
</tr>
</tbody>
</table>

The control mode may be overridden by earlier use of the WHISTCTL keyword.

4. Observed oil production rate

- **UNITS:** sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)
- **DEFAULT:** 0.0

5. Observed water production rate

- **UNITS:** sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)
- **DEFAULT:** 0.0

6. Observed gas production rate

- **UNITS:** sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)
- **DEFAULT:** 0.0

7. Production well VFP table number

   (see keyword VFPROD)

   Set this to zero if you do not require the THP to be calculated.

   If set > 0, the well’s THP is calculated and reported, but the lower limit check is disabled so that the well does not go on to THP control.

   **DEFAULT:** Initially zero, subsequently no change from previous value

8. Artificial lift quantity, for use in calculating the THP

   (see keyword VFPROD)

   This should be defaulted if you do not require the THP to be calculated
UNITS: These depend on what the ALQ was meant to represent when the table was created. Refer to section "Record 6: ALQ values" in the description of keyword VFPPROD.

DEFAULT: Initially zero, subsequently no change from previous value

9. Observed tubing head pressure (THP)

Values entered here are only used for reporting purposes, to compare with the calculated THP. The observed THPs may be written to the SUMMARY file using the SUMMARY section keyword WTHPH. The well does not change to THP control if the calculated THP falls below this value.

Default this item if you have no observed THP value to enter.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

DEFAULT: 0.0

10. Observed bottom hole pressure (BHP)

The observed BHPs may be written to the SUMMARY file using the SUMMARY section keyword WBHPH.

Default this item if you have no observed BHP value to enter.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

DEFAULT: 0.0

ECLIPSE 300 only

11. Observed wet gas production rate

The observed wet gas production rate may be written to the Summary File using the SUMMARY section keyword WWGPRH.

UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: 0.0

ECLIPSE 300 only

12. Observed NGL production rate

The observed NGL production rate may be written to the Summary File using the SUMMARY section keyword WNLPRH.

UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: 0.0

End the data record with a slash (/).

End the set of records with a blank record, containing only a slash.

Note: If items 7 and 8 are defaulted, the VFP table number and ALQ value for the well keep their previous values. Their values are initially set to 0 and 0.0 respectively when the well is first declared a History Matching Well. If you want the THP to be calculated during the history match, set these quantities to the required values either in the well’s first WCONHIST keyword or by subsequently using keyword WELTARG, then subsequent WCONHIST keywords can have their items 7 and 8 defaulted. This saves having to re-specify these items continually in all WCONHIST keywords.
Summary file output

The observed production rates and totals, phase ratios (water cut and GOR for example) and pressures (BHP, THP), of history matching wells can be written to the Summary file, along with their calculated values. The SUMMARY section keywords denoting observed values end with an H (such as WOPRH).

The observed values are set from the data entered in the WCONHIST keyword. Thus the user can construct a graph comparing, for example, the calculated water cut of a history matching well (WWCT) with its observed water cut (WWCTH). The observed rates are also accumulated over time (for example WOPTH), and these may be compared with the calculated cumulative totals (for example WOPT).

The corresponding field and group keywords (for example FOPRH, GWPRH) refer to the sum of their subordinate well flows, with the flows of all history matching wells replaced with their historical rates specified in WCONHIST. Thus, for example, FWCT can be compared with FWCTH to determine how well the field water cut matches its observed value.

If a well is subsequently taken out of history matching mode, by re-specifying it with WCONPROD, the Summary file output is calculated as described below:

- The observed rates and ratios (for example, WOPRH, WWCTH) are set to zero, as there is no observed data input.
- Their cumulative totals are still incremented over time, using the well’s calculated rates. Thus the difference between, for example, WOPTH and WOPT at any time will reflect the cumulative difference between the observed and calculated rates integrated over the period in which the well was in history matching mode.
- The group historical rates (for example, GOPRH) are the sum of the observed rates of their subordinate history matching wells and the calculated rates of any wells that are not in history matching mode, adjusted by any efficiency factors. Thus the difference between, for example, GOPR and GOPRH reflects the difference between the observed rates and calculated rates of all its wells that are currently in history matching mode.
- The group historical cumulative totals (for example GOPTH) are the sum of the well cumulative historical totals (WOPTH), and are also equal to the group historical rates (GOPRH) integrated over time.

Example

```
WCONHIST
  -- well   open/   ctrl   oil   water  gas   VFP   ALQ   obs  obs
  -- name   shut    mode   rate  rate   rate        tab    THP BHP
PROD1  OPEN   LRAT   2410  320    2892 /
PROD2  OPEN   RESV   1970  725    2364 /
PROD3  OPEN   RESV   1507   431   1808   2   0.0  321 2314 /
```


**WCONINJ**  
**Injection well control data, with no group control**

*Attention:* This keyword is obsolete and has been superseded by the simpler keyword **WCONINJE**, which is recommended in place of this one. The retired keyword has been retained for back-compatibility with old datasets and is described in **WCONINJ** in the *ECLIPSE Technical Description*.
Control data for injection wells

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated early with a slash (/), the remaining items taking default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Injector type

   WATER Water injector
   GAS Gas injector

   When running in compositional mode, the nature of the injected gas must also be specified with keyword WINJGAS, unless the well is subordinate to a group with gas injection controls where the injection fluid is defined.

   An alternative gas injector type, HCGAS can be used. The difference between GAS and HCGAS injectors is described in "Calculation of molar rates in ECLIPSE 300"

   Oil injector

   An alternative oil injector type, HCOIL must be used.

   Multiphase injector

   Water and/or gas and/or oil injection.

   Water and/or gas injection. To inject oil and gas, use HCGAS or HCOIL, as described in "Calculation of molar rates in ECLIPSE 300"

   This injects the preferred phase of the well (as specified with the keyword WELSPECS) plus the other phases for which non-zero surface volume proportions are specified in items 12 to 14.

   Gas injector

   When running in compositional mode, the nature of the injected gas must also be specified with keyword WINJGAS.

   An alternative gas injector type, GAS can be used. The difference between GAS and HCGAS injectors is described in "Calculation of molar rates in ECLIPSE 300"
When running in compositional mode, the nature of the injected oil must be specified with keyword WINJOIL.

See "Calculation of molar rates in ECLIPSE 300"

### Thermal

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>STEAM–GAS</td>
<td>Steam and gas injector</td>
</tr>
<tr>
<td>STEAM–OIL</td>
<td>Steam and oil injector</td>
</tr>
</tbody>
</table>

When running in thermal mode, the enthalpy of the injected fluid must be specified with keyword WINJTEMP. If steam and water are injected, the injector type should be defined as WATER and the injection rate (item 5) should be set as the cold water equivalent (C.W.E.) rate. If steam, water and gas are injected, the injector type should be defined as STEAM–GAS and the injection rate (item 5) should be set as the cold water equivalent (C.W.E.) rate of the steam and water; the rate of gas injection can be specified using item 11. If steam, water and oil are injected, the injector type should be defined as STEAM–OIL and the injection rate (item 5) should be set as the cold water equivalent (C.W.E.) rate of the steam and water; the ratio of oil volume to steam volume (C.W.E.) can be specified using item 15.

3. Open/shut flag for the well

   OPEN  Well open for injection.

   STOP  Well stopped off above the formation.

   SHUT  Well completely isolated from the formation

   **ECLIPSE 100 only**

   AUTO  Well initially SHUT, but is opened automatically as soon as constraints on drilling rate (keyword WDRILTIM), drilling rig availability (keyword GRUPRIG), and the maximum number of open wells per group (keyword GECON item 9) allow. If no such constraints are specified, the well opens immediately. Wells on AUTO are opened in the order in which they are first specified in keyword WELSPECS, unless any belong to a group that already has its maximum number of open wells or has all its available drilling rigs already occupied.

   **DEFAULT:** OPEN

4. Control mode

   RATE  Controlled by surface flow rate target (item 5)

   RESV  Controlled by reservoir volume rate target (item 6)

   BHP   Controlled by BHP target (item 7)

   THP   Controlled by THP target (item 8)

   The control mode is not available with multiphase injectors.

   GRUP  The well is immediately under group control, to inject its share of a group or field target set with the keyword GCONINJE.

5. Surface flow rate target or upper limit. User defined arguments can be set with keywords UDQ and UDADIMS.

**Keywords**

<table>
<thead>
<tr>
<th>Reference Manual</th>
<th>Page</th>
<th>Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>ECLIPSE Reference Manual</td>
<td>2448</td>
<td></td>
</tr>
</tbody>
</table>
For a multiphase injector, this is the surface flow rate of the preferred phase of the well.

**UNITS:** \(\text{sm}^3\)/day (METRIC), \(\text{stb}/\text{day}\) for oil or water (FIELD), \(\text{Mscf}/\text{day}\) for gas (FIELD), \(\text{scc}/\text{hr}\) (LAB), \(\text{sm}^3\)/day (PVT–M)

**DEFAULT:** No target or limit

6. Reservoir fluid volume rate target or upper limit. User defined arguments can be set with keywords **UDQ** and **UDADIMS**.

**UNITS:** \(\text{rm}^3\)/day (METRIC), \(\text{rb}/\text{day}\) (FIELD), \(\text{rcc}/\text{hr}\) (LAB), \(\text{rm}^3\)/day (PVT–M)

**DEFAULT:** No target or limit

7. BHP target or upper limit. User defined arguments can be set with keywords **UDQ** and **UDADIMS**.

It is recommended that this quantity should be set to a value not exceeding the highest pressure in the PVT tables, to ensure the tables are not extrapolated in the well. It is best not to default this quantity, as its value is used in calculating the well potential.

**UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT–M)

**ECLIPSE 100**
DEFAULT: 6803 atma, or 1.0E5 psia, or 6891 barsa

**ECLIPSE 300**
DEFAULT: 1000 atma = 14695.9 psia = 1013.25 barsa

**Thermal**
DEFAULT: 200 atma = 2939.1 psia = 202.65 barsa

**E300 CO2STORE**
DEFAULT: 592.15 atma = 8702.18 psia = 600 barsa

All defaults stated are accurate to 2 decimal places.

8. THP target or upper limit. User defined arguments can be set with keywords **UDQ** and **UDADIMS**.

**UNITS:** barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT–M)

**DEFAULT:** No target or limit

9. Injection well VFP table number

(See keyword **VFPINJ**)

Set this to zero if no THP calculations are required and the limiting THP value has been defaulted.

**DEFAULT:** 0

**ECLIPSE 100 only**
This applies unless item 92 of **OPTIONS** is greater than zero, in which case the default is initially zero, and subsequently no change from the previous value. This mimics the behavior of **WCONINJH** item 7.

10. Vaporized oil concentration in the injected gas, or dissolved gas concentration in the injected oil.

This item is ignored for multiphase injectors.

**UNITS:** \(\text{sm}^3/\text{sm}^3\) (METRIC), \(\text{stb}/\text{Mscf}\) for gas injectors (FIELD), \(\text{Mscf}/\text{stb}\) for oil injectors (FIELD), \(\text{scc}/\text{scc}\) (LAB)

**DEFAULT:** 0.0

**Thermal**
11. Ratio of gas volume to steam volume (C.W.E.) for a STEAM–GAS injector. User defined arguments can be set with keywords **UDQ** and **UDADIMS**.
UNITs: sm³/sm³ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm³/sm³ (PVT-M)
DEFAULT: 0.0

12. Surface volume proportion of oil in a multiphase injector.
This must be greater than zero if the preferred phase of the multiphase injection well (set in the keyword WELSPECS) is oil.
UNITs: dimensionless
DEFAULT: 0.0

ECLIPSE 100 only

This must be greater than zero if the preferred phase of the multiphase injection well (set in the keyword WELSPECS) is water.
UNITs: dimensionless
DEFAULT: 0.0

This must be greater than zero if the preferred phase of the multiphase injection well (set in the keyword WELSPECS) is gas.
UNITs: dimensionless
DEFAULT: 0.0

15. Ratio of oil volume to steam volume (C.W.E.) for a STEAM–OIL injector (for use with steam-solvent injection).
UNITs: sm³/sm³ (METRIC), stb/stb (FIELD), scc/scc (LAB), sm³/sm³ (PVT-M)
DEFAULT: 0.0

Thermal

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also keyword WELTARG, which can be used to reset selected control quantities individually.

Calculation of molar rates in ECLIPSE 300

In compositional injection, volume rates are converted to molar rates (for the injector type settings, see item 2 of the WCONINJE keyword):

• For GAS and WATER injectors, a constant molar density is used. The gas molar density is determined from the ideal gas law. The water molar density is determined from the water gravity density specified by the DENSITY or GRAVITY keyword. The molar densities used in a simulation are output to the “Standard conditions” section of the report file.

• For HCGAS and HCOIL injectors, a flash at stock tank conditions (STCOND) is used to calculate the phase molar densities.

Thermal

• For STEAM–GAS and STEAM–OIL injectors, a flash at stock tank conditions (STCOND) is also used to calculate the gas or oil molar densities. For back compatibility with pre-2012.1 versions of STEAM–GAS, set item 276 in OPTIONS3 to 1 to use a fixed gas molar density.
The stock-tank flash used for HCGAS, HCOIL, STEAM–GAS and STEAM–OIL injectors may not produce a single phase fluid of the expected type. For example an HCGAS injection fluid may not flash to single phase gas. Therefore:

- If the flash produces a two-phase fluid, then both phases will be injected and the well will be controlled by the rate of the specified phase. This mechanism can be used to inject two-phase hydrocarbons.
- If the flash produces a single-phase fluid of the wrong type then a constant molar density will be used, determined from

\[ \frac{n}{V} = \frac{P}{RTZ} \]

For HCGAS, a compressibility of \( Z = 1 \) is used when the surface flash finds no gas, giving the same values as the GAS injector molar density. For HCOIL, a fixed compressibility of \( Z = 0.05 \) is used when the surface flash finds no oil.

HCGAS allows different injection streams to have different molar densities. However it should not be used when reservoir gas is reinjected (see WINJGAS), since the injection fluid composition is not known in advance.

In black oil simulations, HCGAS and GAS are the same, and HCOIL is the equivalent to the ECLIPSE 100 OIL injector.

Notes

- The surface proportions in items 12 to 14 are the ratio of the surface volumes of the various phases. Due to the different units used to specify surface volumes for liquids and gas at surface conditions in FIELD units, care must be taken when specifying the surface proportions of multiphase well involving gas. See Example 3 for an illustration.
- When specifying the preferred phase for a multiphase injector, it may help to speed up the convergence of the well model if you specify the phase for which the proportion is greatest as the preferred phase.
- If you have a large proportion of the non-preferred phases, you may have to increase the value of MXWSIT in TUNING keyword record 3 in order to allow the well model to converge. The simulator will give a message if this is the case.

Examples

Example 1

```
WCONINJE
  INJ1 GAS OPEN RATE 500 1* 5000 1000 2 /
  INJ2 WAT OPEN RESV 1* 1000 4000 /
  INJ3 WAT OPEN RATE 700 1* 5000 /
/
```

Example 2

This example shows a multiphase (oil, water and gas) injector, MI1, on which gas has previously been specified as the preferred phase. The units in this example are METRIC.
In this example, a gas rate target of 16,000 sm$^3$/day has been specified, and the proportions of oil to water to gas is 0.05: 0.15: 0.80 (as specified items 12 to 14). Hence the injection fluid will consist of 16,000 sm$^3$/day of gas, (16,000*0.05/0.8 =) 1,000 sm$^3$/day of oil and (16,000*0.15/0.8 =) 3,000 sm$^3$/day of water.

**Example 3**

This example shows a multiphase (water and gas) injector, MI2, on which water has previously been specified as the preferred phase. The units in this example are FIELD.

In this example, a water rate target of 2,000 stb/day has been specified, and the proportion of water to gas is 0.2: 0.8 (as specified items 13 and 14). Hence the injection fluid will consist of 2,000 stb/day of water and (2,000*0.2/0.8 =500 stb/day =) 2.807 Mscf/day of gas.
WCONINJH  Observed rates for history matching injection wells

This keyword is used in place of **WCONINJE** to declare injection wells as special history matching wells, and to enter their observed flow rates (and optionally their measured BHP and THP values). The equivalent keyword for defining history matching production wells is **WCONHIST**.

Wells can be declared as history matching wells during the history matching process, when their injection rates are known. During the simulation these wells are normally be constrained to operate at their observed injection rates, and their calculated BHP and/or THP values are compared against the measured values.

ECLIPSE treats history matching wells differently from ordinary injection wells, in the following ways:

- The observed injection rates entered with this keyword can be written to the Summary file, and also the measured BHP and THP values if they are entered here, for comparison with the rates and pressures calculated during the simulation.

- The set of control modes for the wells is restricted to **RATE** and **BHP**. For **RATE** control the BHP upper limit is automatically set to a large value when the well is first declared a history matching well (the first time it appears in a **WCONINJH** keyword) and when the control mode switches from **BHP** to **RATE**. For **BHP** control the observed BHP is used as the target. This is to reduce the possibility of the well changing to BHP control before its productivity index has been properly matched. (Changing to BHP control at a very high pressure is a sign that the Kh or skin factor is very badly matched.) You can reset the BHP limit to any desired value with the keyword **WELTARG** after the first **WCONINJH** keyword in which the well appears. The well’s BHP limit is not changed by subsequent **WCONINJH** keywords. There is an option (see keyword **WHISTCTL**) to make the run stop automatically if a history matching well changes to BHP control.

**CAUTION:** With multisegment wells (keyword **WELSEGS**), you are recommended to use **WELTARG** after **WCONHIST** to set a BHP limit that is within the range of the PVT tables. If these tables are extrapolated to a very high pressure, it is quite possible that the fluid properties could become unphysical. The multisegment well model requires realistic fluid properties in order to converge. Note that the calculation of the well potential uses the BHP limit, even if the actual operating BHP of the well is far below this value.

The observed flow rates and pressures can be made to change with time by repeating the **WCONINJH** keyword at each timestep. A history matching well can be converted into a normal well (with the standard set of controls) simply by respecifying it with the **WCONINJE** keyword.

The **WCONINJH** keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash.

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the *ECLIPSE Technical Description* for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword **WLIST**.

2. Injector type
   **WATER** Water injector
GAS  Gas injector

ECLIPSE 300 only
When running in compositional mode, the nature of the injected gas must also be specified with keyword WINJGAS, unless the well is subordinate to a group with gas injection controls where the injection fluid is defined.

ECLIPSE 100 only
OIL  Oil injector

MULTI  Multiphase injector (oil and/or water and/or gas).
This injects the preferred phase of the well (as specified with the keyword WELSPECS) plus the other phases for which non-zero surface volume proportions are specified in items 9 to 11.

3. Open/shut flag for the well

OPEN  Well open for injection
STOP  Well stopped off above the formation
SHUT  Well completely isolated from the formation

DEFAULT: OPEN
When the well is shut or stopped, set the observed flow rate to zero. If the flow rates are set to zero while the well is open the behavior depends on whether crossflow is permitted (item 10 in keyword WELSPECS). If crossflow is allowed the well behaves as if stopped (although its status is still reported as OPEN). If crossflow is banned the well will be automatically shut (and its status is reported as SHUT).

4. Observed injection rate (at surface conditions)
For a multiphase injector, this is the surface flow rate of the preferred phase of the well.

UNITS: sm$^3$/day (METRIC), stb/day for oil or water (FIELD), Mscf/day for gas (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: 0.0

5. Observed bottom hole pressure (BHP)
Values entered here are only used for reporting purposes, to compare with the calculated BHP. The observed BHPs may be written to the Summary file using the SUMMARY section keyword WBPH. The well does not change to BHP control if the calculated BHP rises above this value. (The actual BHP upper limit is set to a very high value when the well first appears in a WCONINJH keyword, but you can subsequently reset it to any desired value with the WELTARG keyword.)

Default this item if you have no observed BHP value to enter.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

DEFAULT: 0.0

6. Observed tubing head pressure (THP)
Values entered here are only used for reporting purposes, to compare with the calculated THP. The observed THPs may be written to the Summary file using the SUMMARY section keyword WTHPH. The
well does not change to THP control if the calculated THP rises above this value. Default this item if you have no observed THP value to enter.

UNITs: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

DEFAULT: 0.0

7. Injection well VFP table number

(see keyword VFPINJ)

Set this to zero if you do not require the THP to be calculated.

If set > 0, the well’s THP is calculated and reported, but the upper limit check is disabled so that the well does not go on to THP control.

DEFAULT: Initially zero, subsequently no change from previous value

ECLIPSE 100 only

8. Vaporized oil concentration in the injected gas, or dissolved gas concentration in the injected oil.

This item is ignored for multiphase injectors.

UNITs: sm$^3$/sm$^3$ (METRIC), stb/Mscf for gas injectors (FIELD), Mscf/stb for oil injectors (FIELD), scc/scc (LAB)

DEFAULT: 0.0

ECLIPSE 100 only


This must be greater than zero if the preferred phase of the multiphase injection well (set in the keyword WELSPECS) is oil.

UNITs: dimensionless

DEFAULT: 0.0

10. Surface volume proportion of water in a multiphase injector.

This must be greater than zero if the preferred phase of the multiphase injection well (set in the keyword WELSPECS) is water.

UNITs: dimensionless

DEFAULT: 0.0

11. Surface volume proportion of gas in a multiphase injector.

This must be greater than zero if the preferred phase of the multiphase injection well (set in the keyword WELSPECS) is gas.

UNITs: dimensionless

DEFAULT: 0.0

12. Well control mode

RATE  Well is controlled by injection rate

BHP   Well is controlled by bottom hole pressure

DEFAULT: RATE

End the data record with a slash (/).
End the set of records with a blank record, containing only a slash.

**Notes**

*ECLIPSE 300 only*

In multiphase compositional injection, all hydrocarbons are specified as gas regardless of their phase, and the actual liquid-vapor composition is determined via a flash calculation. Also with this simulator, oil injection is not available in black oil models. Hence the preferred phase for a multiphase injection well (specified in the **WELSPECS** keyword) cannot be set to **OIL**, and the ‘Surface volume proportion of oil in a multiphase injector’ (item 9) is ignored.

**Summary file output**

There are equivalent facilities for writing injection well historical rates and pressures to the **SUMMARY** file as there are for production wells (see keyword **WCONHIST**).

The observed injection rates and their cumulative totals may be written to the **SUMMARY** file using the corresponding **SUMMARY** section keywords ending with an **H** (**WWIRH**, **WGIRH**, **WWITH**, **WGITH**). The observed BHP and THP values, if they are supplied here, may be written to the Summary file using the keywords **WBHPH** and **WTHPH** respectively.

The corresponding field and group keywords (for example **FWIRH**, **GGIRH**) refer to the sum of their subordinate injection wells’ observed, and their cumulative totals.

If a well is subsequently taken out of history matching mode, by re-specifying it with **WCONINJE**, the Summary file output is calculated as described below:

- The observed rates and ratios (for example, **WWIRH**, **WWCTH**) are set to zero, as there is no observed data input.
- Their cumulative totals are still incremented over time, using the well’s calculated rates. Thus the difference between, for example, **WWITH** and **WWIT** at any time will reflect the cumulative difference between the observed and calculated rates integrated over the period in which the well was in history matching mode.
- The group historical rates (for example, **GWIRH**) are the sum of the observed rates of their subordinate history matching wells and the calculated rates of any wells that are not in history matching mode, adjusted by any efficiency factors. Thus the difference between, for example, **GWIR** and **GWIRH** reflects the difference between the observed rates and calculated rates of all its wells that are currently in history matching mode.
- The group historical cumulative totals (for example **GWITH**) are the sum of the well cumulative historical totals (**WWITH**), and are also equal to the group historical rates (**GWIRH**) integrated over time.

**Example**

```
WCONINJH
  -- well    well  open/   inj  obs  obs  VFP  Rs/Rv
  -- name    type  shut  rate  BHP  THP  tab
  WINJ    WAT   OPEN  2410  /   
  GINJ    GAS   OPEN  3970  3676  2852  2  / 
/```

**Keywords**

2456
**Other**

See the examples in the `WCONINJE` keyword for an illustration of how to specify the oil, water and gas surface volume proportions for multiphase injectors (`WCONINJH` items 9-11).
**WCONINJP  Control data for pattern flood injection wells**

This keyword is used to specify a well that injects at a reservoir volume rate defined by the production of the wells that surround it. An example of this type of control would be to specify a well that injects at a rate equal to the sum of one quarter of the voidage from each of the four neighboring producers in a five spot pattern.

The keyword can only be used on one injection well at a time, so it must be specified more than once if there is more than one injection well under this type of control.

The keyword is followed by a data record that specifies the injection well and then a number of data records specifying the production wells in the surrounding pattern that each make a contribution to defining the injection well’s rate. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

The data items in each record are:

**Record 1**

This record defines control data for the injection well. It is similar to the **WCONINJE** keyword except that the injection rate is not specified; this rate is calculated at each timestep from the produced voidage of the surrounding pattern of wells that are defined in subsequent records of this keyword.

The record may be terminated early with a slash (/), the remaining items taking default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

1. Name of the pattern flood injection well
2. Injector type
   - WATER  Water injector
   - GAS     Gas injector
   - OIL     Oil injector
   - MULTI   Multiphase injector (oil and/or water and/or gas).

   *ECLIPSE 300*  When running in compositional mode, the nature of the injected gas must also be specified with keyword **WINJGAS**.

   *ECLIPSE 100 only*  This injects the preferred phase of the well (as specified with the keyword **WELSPECS**) plus the other phases for which non-zero surface volume proportions are specified in items 9 to 11.

3. Open/shut flag for the well
   - OPEN  Well open for injection.
   - STOP  Well stopped off above the formation.
   - SHUT  Well completely isolated from the formation

   DEFAULT: OPEN

4. BHP upper limit

---

**Keywords**
It is recommended that this quantity should be set to a value not exceeding the highest pressure in the PVT tables, to ensure the tables are not extrapolated in the well. It is best not to default this quantity, as its value is used in calculating the well potential.

UNITs: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

**ECLIPSE 100**
DEFAULT: 6804 atma, or 1.0E5 psia, or 6895 barsa

**ECLIPSE 300**
DEFAULT: 1000 atma = 14700 psia = 1013 barsa

**Thermal**
DEFAULT: 200 atma = 2940 psia = 202.6 barsa

**E 300 CO2STORE**
DEFAULT: 592.15 atma=8702 psia =600 barsa

5. THP upper limit
UNITs: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
DEFAULT: No limit

6. Injection well VFP table number
(See keyword **VFPINJ**)
Set this to zero if no THP calculations are required and the limiting THP value has been defaulted.
DEFAULT: 0

**ECLIPSE 100 only**
7. Vaporized oil concentration in the injected gas, or dissolved gas concentration in the injected oil
This item is ignored for multiphase injectors.
UNITs: sm$^3$/sm$^3$ (METRIC), stb/Mscf for gas injectors (FIELD), Mscf/stb for oil injectors (FIELD), scc/scc (LAB)
DEFAULT: 0.0

8. Injection voidage rate target multiplier
This multiplies the calculated voidage rate target for the pattern flood injector.
DEFAULT: 1.0

**ECLIPSE 100 only**
This must be greater than zero if the preferred phase of the multiphase injection well (set in the keyword **WELSPECS**) is oil.
UNITs: dimensionless
DEFAULT: 0.0

10. Surface volume proportion of water in a multiphase injector.
This must be greater than zero if the preferred phase of the multiphase injection well (set in the keyword **WELSPECS**) is water.
UNITs: dimensionless
DEFAULT: 0.0

11. Surface volume proportion of gas in a multiphase injector.
This must be greater than zero if the preferred phase of the multiphase injection well (set in the keyword WELSPECS) is gas.

UNITs: dimensionless

DEFAULT: 0.0

End data record with a slash (/).

Each subsequent record

The subsequent records each define a production well and the fraction of its reservoir volume rate that contributes to the overall rate for the injection well specified in record 1.

1. Production well name, well name template, well list or well list template

A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. The production rate fraction for the well(s) in item 1 of this record

This fraction multiplies the well’s reservoir volume rate (or part thereof, see item 3 below). This product, when added to those from all the other contributing producers, defines the flow rate for the injection well specified in record 1.

DEFAULT: The fraction will be updated at the beginning of each timestep on the basis of the number of injectors assigned to replace the voidage of each producer. A producer’s fractions will thus change if one of its associated injectors is shut (or opened), with the current set of open injectors effectively automatically compensating to ensure the producer’s voidage is still replaced.

3. Fluid-in-place region number

If this is specified, then the voidage rate contribution from this well is restricted to the sum of the voidage flows from connections to grid cells that fall within this fluid-in-place region. Note that connection voidage rates are calculated at the reference pressures of the appropriate fluid-in-place regions; these pressures may differ from the reference pressure used to calculate the well’s overall voidage rate. (See item 88 in OPTIONS3 or item 85 in OPTIONS.)

DEFAULT: 0

End data record with a slash (/).

Notes

• A pattern flood injection well is automatically set as unavailable for higher level group control.

• If an injector has previously been defined as a pattern flood injector and is then re-specified using this keyword without any associated production wells, it will be switched back to being a normal injector. This switch may also be achieved by re-specifying the control mode of the well using the WCONINJE keyword.

• The control mode of a pattern flood injector is reported as ‘PFVR’ in the print file.

• Although there is no limit to the number of producers that may contribute to the overall rate of a pattern flood injector, the voidage from a particular producer cannot be split between more than ten pattern flood injectors.

ECLIPSE 300 only

Keywords

2460
• In multiphase compositional injection, all hydrocarbons are specified as gas regardless of their phase, and the actual liquid-vapor composition is determined via a flash calculation. Also with this simulator, oil injection is not available in black oil models. Hence the preferred phase for a multiphase injection well (specified in the \textbf{WELSPECS} keyword) cannot be set to \textbf{OIL}, and the ‘Surface volume proportion of oil in a multiphase injector’ (item 9) is ignored.

\textbf{Example}

The water injection well INJW replaces a quarter of the reservoir voidage of each of its four surrounding producers. This is effectively its share of the production voidage in a repeated five spot pattern.

\begin{verbatim}
WCONINJP
INJW WAT 1* 5000 /
PRD1 /
PRD2 /
PRD3 /
PRD4 /
\end{verbatim}

\textbf{Other}

See the examples in the \textbf{WCONINJE} keyword for an illustration of how to specify the oil, water and gas surface volume proportions for multiphase injectors (\textbf{WCONINJP} record 1, items 9-11).
WCONPROD  Control data for production wells

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated early with a slash (/). The remaining items will assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Open/shut flag for the well
   
   OPEN  Well open for production
   STOP  Well stopped off above the formation
   SHUT  Well completely isolated from the formation
   AUTO  Well initially SHUT, but is opened automatically as soon as constraints on drilling rate (keyword WDRILTIM), drilling rig availability (keyword GRUPRIG), and the maximum number of open wells per group (keyword GECON item 9) allow. If no such constraints are specified, the well will open immediately. Wells on AUTO are opened in the order in which they are first specified in keyword WELSPECS, unless any belong to a group that already has its maximum number of open wells or has all its available drilling rigs already occupied.
   
   It is preferable to shut or stop the well rather than give it a zero flow rate target. In particular, a zero flow rate target applied to a well with a THP limit may cause hunting between control modes. Furthermore, ECLIPSE 300 interprets a zero rate limit (applied to any item that is not the control mode's target) as a defaulted, hence inactive, limit.
   
   DEFAULT: OPEN

3. Control mode
   
   ORAT  Controlled by oil rate target (item 4)
   WRAT  Controlled by water rate target (item 5)
   GRAT  Controlled by gas rate target (item 6)
   LRAT  Controlled by liquid rate target (item 7)
   CRAT  Controlled by linearly combined rate target (item 19)
   RESV  Controlled by reservoir fluid volume rate target (item 8)
   BHP   Controlled by BHP target (item 9)
THP Controlled by THP target (item 10)

WGRA Controlled by wet gas rate target (item 13)

**ECLIPSE 300 only**
TMRA Controlled by total molar rate target (item 14)

**ECLIPSE 300 only**
STRA Controlled by steam rate target (item 15) (available only with the Thermal option)

**ECLIPSE 300 only**
SATP Controlled by water saturation pressure (item 16) (available only with the Thermal option)

**ECLIPSE 300 only**
SATT Controlled by water saturation temperature (item 17) (available only with the Thermal option)

**ECLIPSE 300 only**
CVAL Controlled by calorific rate target (item 18). Molar calorific values must be specified with the CVAL keyword before this control mode may be used.

**ECLIPSE 300 only**
NGL Controlled by NGL rate target (item 20).

GRUP The well is immediately under group control, to produce its share of a group or field target rate set with keyword GCONPROD.

DEFAULT: ‘ ’ (Undefined)

**ECLIPSE 100**
The control mode can be left defaulted if the well is shut/stopped and is not placed in the drilling queue.

4. Oil rate target or upper limit. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: No target or limit

5. Water rate target or upper limit. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: No target or limit

6. Gas rate target or upper limit. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

**ECLIPSE 300**
In compositional mode, the gas rate of a well is defined by passing the produced well hydrocarbon through a copy of the separator to which the well is assigned, or by a flash to standard conditions if no separator has been defined.

UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: No target or limit

7. Liquid rate target or upper limit. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: No target or limit
8. Reservoir fluid volume rate target or upper limit. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

The reservoir fluid volume rate is the volume of the fluids produced when measured at the average hydrocarbon pressure in the field, or the average pressure in a nominated fluid-in-place region if item 13 is set in WELSPECS.

UNITS: $\text{m}^3$/day (METRIC), rb/day (FIELD), rcc/hr (LAB), $\text{m}^3$/day (PVT-M)

DEFAULT: No target or limit

9. BHP target or lower limit. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

ECLIPSE 100
DEFAULT: 1 atma = 14.70 psia

ECLIPSE 300
DEFAULT: 100 atma = 1469.59 psia

If the control mode in item 3 is put to BHP, then this item cannot be defaulted. It is recommended that this quantity should be set to a value not less than the lowest pressure in the PVT tables, to ensure the tables are not extrapolated in the well. It is best not to default this quantity, as its value is used in calculating the well potential. Further, if a well approaches the default BHP, calculated THP values will become negative in ECLIPSE 100.

10. THP target or lower limit. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

DEFAULT: 0.0

ECLIPSE 100
A THP limit of zero will prevent the well switching to THP control, whatever the value of its calculated THP. But its THP is still calculated, provided that it has a positive VFP table number in item 11.

11. Production well VFP table number (see keyword VFPPROD)

Set this to zero if no THP calculations are required and the limiting THP value has been set to zero.

DEFAULT: 0

12. Artificial lift quantity, for use in THP calculations (see keyword VFPPROD)

UNITS: These depend on what the ALQ was meant to represent when the table was created. Refer to section "Record 6: ALQ values" in the description of keyword VFPPROD.

DEFAULT: 0.0

ECLIPSE 300 only

13. Wet gas production rate target or upper limit

Wet gas rate is related to the hydrocarbon moles produced using a unity Z-factor at surface conditions.

UNITS: $\text{m}^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), $\text{m}^3$/day (PVT-M)

DEFAULT: No target or limit

ECLIPSE 300 only

14. Total molar rate target or upper limit. (Note that the total molar rate includes water.)

UNITS: kg-M/day (METRIC), lb-M/day (FIELD), gm-M/hr (LAB), kg-M/day (PVT-M)
15. Steam production rate target or upper limit, Cold Water Equivalent. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS. (available only with the Thermal option)

UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: No target or limit

16. The pressure offset $\Delta P$ for saturation pressure control. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS. (Available only with the Thermal option.)

The bottom hole pressure is constrained by $BHP \geq Psat + \Delta P$ where $Psat$ is the maximum saturated water pressure in all the producing completion cells. The pressure offset can be output using the WPOFF summary keyword.

UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

17. The temperature offset $\Delta T$ for saturation temperature control. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS (available only with the Thermal option).

The bottom hole pressure is constrained by $BHP \geq Psat (T + \Delta T)$ where $T$ is the maximum temperature in all the producing completion cells. The temperature offset can be output using the WTOFF summary keyword.

UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

18. Calorific rate target or upper limit

UNITS: kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB), kJ/day (PVT-M)

DEFAULT: No target or limit

19. Linearly combined rate target or upper limit. Linear combination coefficients must be set with keyword LINCOM. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

UNITS: determined by keyword LCUNIT

DEFAULT: No target or limit

20. NGL rate target or upper limit. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: No target or limit

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also keyword WELTARG, which can be used to reset selected control quantities individually.

**Example**

```
WCONPROD
PROD3 OPEN LRAT 2*      5000 1000 1* 2000 100 1 0.0 /
```
**WCUTBACK  Well cutback limits (for producers and injectors)**

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

Default values can be specified by null repeat counts of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword `WLIST`.

2. Water cut upper limit for well rate cutback (producers only)
   
   A value of 0.0 switches off this limit
   
   **UNITS:** sm³/sm³ (METRIC), stb/stb (FIELD), scc/scc (LAB), sm³/sm³ (PVT-M)
   
   **DEFAULT:** No limit

3. Gas-oil ratio upper limit for well rate cutback (producers only)
   
   A value of 0.0 switches off this limit
   
   **UNITS:** sm³/sm³ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm³/sm³ (PVT-M)
   
   **DEFAULT:** No limit

4. Gas-liquid ratio upper limit for well rate cutback (producers only)
   
   A value of 0.0 switches off this limit
   
   **UNITS:** sm³/sm³ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm³/sm³ (PVT-M)
   
   **DEFAULT:** No limit

5. Water-gas ratio upper limit for well rate cutback (producers only)
   
   A value of 0.0 switches off this limit
   
   **UNITS:** sm³/sm³ (METRIC), stb/Mscf (FIELD), scc/scc (LAB), sm³/sm³ (PVT-M)
   
   **DEFAULT:** No limit

6. Rate cutback factor (fraction)
   
   This is the ratio of the reduced flow rate to the current flow rate. It should be less than 1.0

   **ECLIPSE 100** The same rate cutback factor applies to both this keyword and the `WBHGLR` keyword. If `WBHGLR` has been entered previously, this item overwrites item 4 in that keyword. If `WBHGLR` is entered subsequently, its item 4 overwrites this item.

   **ECLIPSE 100** **DEFAULT:** same as item 4 in the `WBHGLR` keyword if that has been entered previously; otherwise 1.0.

7. Control phase, to which the reduced rate applies
OIL  Reduced oil rate

WAT  Reduced water rate

GAS  Reduced gas rate

LIQ  Reduced liquid rate

RESV Reduced reservoir fluid volume rate

COMB Reduced linearly combined rate (coefficients must be set with LINCOM)

ECLIPSE 300 only

WGAS Reduced wet gas rate

ECLIPSE 300 only

NGL Reduced NGL rate

ECLIPSE 100

You are advised to choose a phase that does not have a user defined rate limit, as it would be overwritten whenever a cutback or cutback reversal occurs; see Note 3 below.

ECLIPSE 100

The same control phase applies to both this keyword and the WBHGLR keyword. If WBHGLR has been entered previously this item overwrites item 5 in that keyword. If WBHGLR is entered subsequently its item 5 overwrites this item.

ECLIPSE 100

DEFAULT: same as item 5 in the WBHGLR keyword if that has been entered previously; otherwise, no default is allowed.

8. Grid block pressure limit for well rate cutback
   (Applied to any block containing an open connection to the well)
   A value of 0.0 or > 1.0E19 switches off this limit.
   This is treated as a lower limit for production wells, and an upper limit for injection wells.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
   DEFAULT: 0.0

9. Grid block pressure limit for reversing the cutback process
   (Applied to any block containing an open connection to the well.)
   A value of 0.0 or > 1.0E19 switches off this limit.
   This is treated as an upper limit for production wells, and a lower limit for injection wells.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
   DEFAULT: 0.0

10. Water cut lower limit for reversing the cutback process (producers only)
    A value of 0.0 switches off this limit.
    UNITS: sm³/sm³ (METRIC), stb/stb (FIELD), scc/scc (LAB), sm³/sm³ (PVT-M)
    DEFAULT: 0.0

11. Gas-oil ratio lower limit for reversing the cutback process (producers only)
A value of 0.0 switches off this limit.

UNITs: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)

DEFAULT: 0.0

12. Gas-liquid ratio lower limit for reversing the cutback process (producers only)
A value of 0.0 switches off this limit.

UNITs: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)

DEFAULT: 0.0

13. Water-gas ratio lower limit for reversing the cutback process (producers only)
A value of 0.0 switches off this limit.

UNITs: sm$^3$/sm$^3$ (METRIC), stb/Mscf (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)

DEFAULT: 0.0

14. Remove cutbacks when the well is worked over?

YES  When the well is worked over after violating an economic limit (set in keywords CECON and WECON for example), any cutbacks performed on the well are completely reversed by removing the rate limit on the control phase specified in item 7.

NO   Any cutbacks made to the well remain in force (unless reversed by the application of cutback reversal limits).

ECLIPSE 100

The instruction applies to both this keyword and the WBHGLR keyword. If WBHGLR has been entered previously, this item overwrites item 8 in that keyword. If WBHGLR is entered subsequently, its item 8 overwrites this item.

DEFAULT: NO

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Notes

1. Whenever the well violates one of the cutback limits given in items 2 to 5 and 8, it is set to operate at a reduced target rate of the phase specified in item 7. Its control mode will automatically change to the mode corresponding to item 7 (if it is not already under that mode of control). The target rate of the control phase is set equal to the well’s current flow rate of that phase, multiplied by the cutback factor specified in item 6.

2. Whenever the well violates one of the cutback reversal limits given in items 9 to 13, and if the well is currently operating under rate control of the phase selected in item 7, the target rate of the phase is increased by dividing it by the factor specified in item 6. This essentially reverses the cutback process.

3. The well’s target rate of the phase selected in item 7 is reset each time one of the limits is violated. Any initially user-defined rate limit for the phase is ‘forgotten’. Thus a series of cutback reversals could increase the rate target for that phase beyond its initial limit. This can be avoided by choosing a phase in item 7 for which there is no user-defined limit. For example, if the well has rate limits defined directly for just the oil and water phases, a suitable control phase would be LIQ or RES. The well
changes to this control mode when it is cut back. During cutback reversals this limit is progressively increased, but the original oil and water limits remain intact and are not exceeded.

4. The keyword can be applied to both producers and injectors. The limits in items 2 to 5 and items 10 to 13 refer to production wells only. The pressure limits in items 8 and 9 can be applied to both production and injection wells.

See also keyword \texttt{WLIMTOL}.

**Example**

```
WCUTBACK
 'P*'  0.6  3*  0.9  LIQ  3000 /
 WELL2  1*  3.0  2*  0.9  RESV  3*  2.0 /
```
WCUTBACT  Tracer cutback limits for production wells

WCUTBACT is followed by any number of records, described below, each record terminated by a slash (/). A record may be terminated early at any item; the remaining items assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted. There are two types of record; one describes the connection and the second lists the tracers and their limits.

The set of records must end with a blank record, containing only a slash (/).

Records for the connection

Records of the first kind can contain some or all of the following items:

1. Well name, well name template, well list or well list template
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Rate cutback factor (fraction)
   
   This is the ratio of the reduced flow rate to the current flow rate. It should be less than 1.0.

   The same rate cutback factor applies to both this keyword and the WBHGLR / WCUTBACK keyword. If WBHGLR or WCUTBACK has been entered previously, this item overwrites item 4 in WBHGLR or item 6 in WCUTBACK. If WBHGLR or WCUTBACK is entered subsequently, item 4 of WBHGLR or item 6 of WCUTBACK overwrites this item.

   DEFAULT: same as item 4 in the WBHGLR keyword or item 6 in the WCUTBACK keyword, if entered previously; otherwise 1.0.

3. Control phase, to which the reduced rate applies
   
   OIL  Reduced oil rate
   WAT  Reduced water rate
   GAS  Reduced gas rate
   LIQ  Reduced liquid rate
   RESV Reduced reservoir fluid volume rate
   COMB Reduced linearly combined rate (coefficients must be set with LINCOM)

4. Remove cutbacks when the well is worked over?
   
   YES  When the well is worked over after violating an economic limit (set in keywords CECON and WECON for example), any cutbacks performed on the well are completely reversed by removing the rate limit on the control phase specified in item 7.
   
   NO   Any cutbacks made to the well remain in force, unless reversed by application of cutback reversal limits.

   End the record with a slash (/)
Subsequent records for the tracers

There can follow a maximum of three subsequent records for each well. These records have some or all of the following items.

1. Tracer name (Use PLY for a polymer and BRI for brine)
2. Total tracer rate upper limit for well rate cutback
   DEFAULT: No limit
3. Total tracer rate lower limit for well rate cutback reversal
   DEFAULT: No limit
4. Total tracer concentration upper limit for well rate cutback (not available for partitioned tracers)
   DEFAULT: No limit
5. Total tracer concentration lower limit for well rate cutback reversal (not available for partitioned tracers)
   DEFAULT: No limit

End the record with a slash (/).

End the data for this well with a blank line containing just a slash (/).

Note that this means that the keyword will end with two blank lines containing just slashes.

Notes

1. No units are given for the tracer rates. The rates are those of the carrier flow.
2. Since the tracers are solved after the main reservoir simulation if it is subsequently discovered that a limit is violated, the timestep is always repeated with the action enforced from the start of the timestep. Even with this it may be that the ‘action’ lags the violation by a timestep.
3. Whenever the well violates one of the cutback limits given in items 2 to 5 of record 2, it is set to operate at a reduced target rate of the phase specified in item 3 of record 1. Its control mode will automatically change to the mode corresponding to item 3 (if it is not already under that mode of control). The target rate of the control phase is set equal to the well’s current flow rate of that phase, multiplied by the cutback factor specified in item 2.
4. Whenever the well violates one of the cutback reversal limits, and if the well is currently operating under rate control of the phase selected in item 3 of record 1, the target rate of the phase is increased by dividing it by the factor specified in item 2 of record 1. This essentially reverses the cutback process.
5. The well’s target rate of the phase selected in item 3 of record 1 is reset each time one of the limits is violated. Any initially user-defined rate limit for the phase is ‘forgotten’. Thus a series of cutback reversals could increase the rate target for that phase beyond its initial limit. This can be avoided by choosing a phase in item 3 of record 1 for which there is no user-defined limit. For example, if the well has rate limits defined directly for just the oil and water phases, a suitable control phase would be LIQ or RESV. The well changes to this control mode when it is cut back. During cutback reversals this limit is progressively increased, but the original oil and water limits remain intact and are not exceeded.
6. This keyword must end with two lines containing only slashes. See the example.
Example

WCUTBACT
  PROD1 0.8 WAT /
  TR1 1000.0 3*/
  TR2 2* 0.5 0.001 /
  TR3 1500.0 10.0 0.5 /
 /
PROD2 0.95 RESV /
  TR3 110.0 90.0 /
 /
PROD3 0.95 RESV /
  TR4 2* 0.6 /
  PLY 2000 3*/
/

Keywords

2473
WCYCLE

Automatic cycling of wells on and off

This keyword provides a means of automatically cycling wells on and off for specified intervals of time. The facility can be used, for example, to simulate a WAG process, where the injection well is represented by separate water and gas injectors that are cycled on and off alternately.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template.
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. On-period - The well is turned off at the beginning of the first timestep that starts after the specified on-period has elapsed since it was last turned on.
   A zero or negative value prevents the well being turned on again by automatic cycling.
   UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
   DEFAULT: 0.0

3. Off-period - If the well was turned off by automatic cycling, it is turned on again at the beginning of the first timestep that starts after the specified off-period has elapsed since it was turned off.
   If the well was closed for any other reason (for example manually or due to an economic limit), it remains closed until action is taken to open it again. Once opened again, it resumes cycling.
   A zero or negative value prevents the well being turned off again by automatic cycling.
   UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
   DEFAULT: 0.0

4. Start-up time - The well’s efficiency factor is steadily ramped up over the specified start-up time when it is turned on by automatic cycling. During each timestep after reopening, its efficiency factor is multiplied by the quantity
   \[
   \frac{(T-T_o)}{\text{Start-up Time}} \quad \text{Eq. 3.219}
   \]
   where
   \[ T \] is the time at the end of the timestep
   \[ T_o \] is the time the well was turned on for as long as \( T-T_o < \text{Start-up Time} \).
   Thus if the start-up time is large compared to the timestep size, the well will be brought on gradually, avoiding the timestep convergence difficulties sometimes observed when a high-rate well is suddenly opened.
   The start-up time also applies to wells turned on by periodic testing (see keyword WTEST).
   UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)
   DEFAULT: 0.0, which gives an instantaneous start-up.
5. Maximum length of the timestep in which the well is turned on by automatic cycling. The value specified here overrides the normal maximum step length (see keyword TUNING) during the timestep in which the well is turned on by automatic cycling. This provides a means of cutting back the timestep when a well is turned on, to avoid the convergence difficulties sometimes observed when a high-rate well is suddenly opened.

A zero or negative value applies no restriction on the timestep size, other than the normal maximum step length.

UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

DEFAULT: 0.0 (no timestep restriction on start-up)

6. Should the timesteps be controlled to coincide exactly with the on/off cycling times of this well?

YES The timesteps coincide with this well’s on/off cycling times, and the well therefore cycles with exactly the specified periods. (If there are several wells cycling on and off at different times, this could slow the run down by imposing small timesteps.)

NO The timesteps does not automatically coincide with this well’s on/off cycling times. The well cycles at the beginning of the next timestep that starts after the specified period has elapsed. The well’s on/off periods may therefore be longer than the values specified, unless they coincide with reporting times.

DEFAULT: NO

End the data record with a slash (/).

End the set of records with a blank record containing just a slash.

Example

To initiate a WAG injection scheme, injecting water for 60 day periods and gas for 40 day periods, starting with water, with a 10 day maximum timestep at each changeover:

Set the cycle periods:

| WCYCLE | -- well on      off     startup  max t/s      control |
|--------|----------------|-----------------|-----------------|-----------------|
|        | -- name period period time on startup timesteps? |
|        | INJW 60.0 40.0 1* 10.0 YES / |
|        | INJG 40.0 60.0 1* 10.0 YES / |

Start with the water injector open and the gas injector shut:

| WEOPEN | INJW OPEN / |
|        | INJG SHUT / |

Advance to the start of the first gas injection period and open the gas injector. It will start cycling:

| TSTEP  | 60.0 / |
| WEOPEN | INJG OPEN / |
WDFAC

Sets well D-factor (flow-dependent skin for gas)

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template.

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. D-factor for the well(s).

   UNITS: day/sm³ (METRIC), day/Mscf (FIELD), hr/scc (LAB), day/sm³ (PVT-M)

   DEFAULT: 0.0

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

The D-factors for wells not specified with this keyword are defaulted to zero, unless they are set in keyword COMPDAT or COMPDATL.

Note: For ECLIPSE 100, if the VDFLOW or the VDFLOWR keyword has been specified and has been applied to the well connections by setting item 218 of the OPTIONS keyword to 2, this will be superseded by the calculation due to the WDFAC keyword.

Converting the well D-factor into the connection D-factors

The D-factor is normally obtained from measurements on the well, and is multiplied by the well’s free gas flow rate to give the non-Darcy flow addition to the skin factor:

\[ S \rightarrow S + D_w \left| Q_{fg} \right| \]

Eq. 3.220

Within the simulator, however, it is treated as a quantity associated with each well connection rather than the well as a whole, and it is multiplied by the connection’s free gas flow rate to give the local non-Darcy skin contribution:

\[ S \rightarrow S + D \left( j \right) \left| q_{fg} \left( j \right) \right| \]

Eq. 3.221

If the well has more than one connection, the connections’ D-factors must be scaled to apply to the connection flows instead of the well flow, in order to give the same addition to the skin factor. ECLIPSE performs the scaling automatically, when the well D-factor is specified. The scaling calculation assumes the free gas flow rate in each connection is proportional to the connection transmissibility factor. It aims to give each connection the same non-Darcy skin, by setting the D-factors in inverse proportion to the connection transmissibility factor:

\[ D \left( j \right) = D_w \frac{\sum T_w \left( i \right)}{T_w \left( j \right)} \]

Eq. 3.222

where
$T_{w,j}$ represents the transmissibility factor for the connection $j$, and

$\sum_{i} T_{w}(i)$ represents the sum of the transmissibility factors of the well’s currently open connections.

Note that the scaling is performed according to the number of connections open at the time \texttt{WDFAC} is entered. It is assumed that the D-factor entered has been measured at this time, while the well has its current number of open connections. The scaling does not of course change the D-factor if the well has only one open connection. If you wish to examine the resulting connection D-factors, request a printout of the Schedule section keyword data, and the D-factors are printed in the table of well connection data.

If subsequently one or more connections in the well are closed, the well’s D-factor changes. However, the D-factors of the remaining connections should not be affected. There is therefore no need to specify the well’s D-factor again, because the connection D-factors by default retain their original scaled values.

If you wish to enter the connection D-factors directly, bypassing the scaling calculation, they may be entered with a negative sign in item 12 of keyword \texttt{COMPDAT} or item 13 of keyword \texttt{COMPDATL}.

See also "The flow-dependent skin factor" in the \textit{ECLIPSE Technical Description} for further discussion on the treatment of flow-dependent skin.

\textbf{Example}

```
WDFAC
-- well   well
-- name   D-factor
  PROD1  1.5E-3  /
  PROD2  1.2E-3  /
/
```

\textit{Note:} If non-Darcy Forchheimer effects for gas flow have been enabled via item 4 of the \texttt{VELDEP} keyword, then a D-factor for each production well connection will be calculated. These calculated, Forchheimer parameter based D-Factors will override those set in this keyword. See "Forchheimer parameter based well D-factors" in the \textit{ECLIPSE Technical Description} for further details.
WDFACCOR  D-factor correlation (flow-dependent skin factor for gas)

The keyword defines an expression for connection D-factors that is based on a correlation for the coefficient of inertial resistance, usually known as $\beta$, in Forchheimer’s flow equation as described below. The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template.
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword \texttt{WLIST}.

2. Coefficient A in the D-factor correlation
   DEFAULT: 0.0

3. Power B of a connected grid block permeability in the D-factor correlation
   DEFAULT: 0.0

4. Power C of a connected grid block porosity in the D-factor correlation
   DEFAULT: 0.0

For each connection in the well, a D-factor will be calculated based on the permeability and porosity of the connected grid block together with the fluid properties of the wellbore. These D-Factors are recalculated each time a \texttt{COMPDAT} or \texttt{COMPDATL} or \texttt{WDFAC} or \texttt{WDFACCOR} is found in the schedule.

The calculated D-factor may be written to the summary file using the keyword \texttt{CDFAC}.

\textbf{Note:} For ECLIPSE 100, if the \texttt{VDFLOW} or the \texttt{VDFLOWR} keyword has been specified and has been applied to the well connections by setting item 218 of the \texttt{OPTIONS} keyword to 2, this will be superseded by the calculation due to the \texttt{WDFACORR} keyword.

The D-factor correlation

Following Dake [Ref. 30], the expression for the non-Darcy flow D-factor used here is:

\[
D = A \cdot K_e^B \cdot \phi^C \cdot \frac{K_e}{h} \cdot \frac{1}{r_w} \cdot \frac{\gamma_g}{\mu_g} \tag{Eq. 3.223}
\]

where

- $A, B, C$ are supplied by this keyword
- $K_e$ is the effective permeability of the connected grid block. For a vertical well the permeability used here is the geometric mean of the $x$- and $y$-direction permeabilities, $K_e = (k_x k_y)^{1/2}$.
- $\phi$ is the porosity of the connected grid block
- $h$ length of the connection
$r_w$ wellbore radius

$\gamma_g$ relative density of produced or injected gas at surface conditions with respect to air at standard temperature and pressure

$\mu_g$ gas viscosity at bottom hole pressure

**ECLIPSE 300**

Note that $\gamma_g$ and $\mu_g$ vary with time and thus so will the calculated D-factor.

**ECLIPSE 100**

Note that $\mu_g$ varies with time and thus so will the calculated D-factor.

See also "The flow-dependent skin factor" in the *ECLIPSE Technical Description* for further discussion on the treatment of flow-dependent skin.

### The coefficient of inertial resistance

The expression for the non-Darcy D-factor can be written as

$$D = \alpha \cdot \beta \cdot \frac{K_e}{h} \cdot \frac{1}{r_w} \cdot \frac{\gamma_g}{\mu_g}$$

where $\beta$ is the coefficient of inertial resistance, given by

$$\beta = \text{const} \cdot K_e^B \cdot \phi^C$$

$\beta$ has Forchheimer units. A laboratory determined relationship for $\beta$ can be obtained by using a $\text{const} = 883.90$ (as converted from Dake), $B = -1.1045$ and $C = 0$ (from Dake).

$\alpha$ is a constant, the value of which depends on the units system, as shown in the see the following table.

<table>
<thead>
<tr>
<th>Metric</th>
<th>Field</th>
<th>Lab</th>
<th>PVT-Metric</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unit</td>
<td>day / sm$^3$</td>
<td>day / Mcf</td>
<td>hr / scc</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>$2.24460 \times 10^{-10}$</td>
<td>$6.83352 \times 10^{8}$</td>
<td>$5.41375 \times 10^{11}$</td>
</tr>
</tbody>
</table>

*Table 3.51: Unit-dependent constant for the Forchheimer parameter-dependent skin-factor*

Thus Dake’s formula (in field units) can be specified by entering $A = 6.04e \cdot 5$, $B = -1.1045$ and $C = 0$.

#### Example

```
WDFACCOR
-- well Coeff Power Power
-- name A B C
  PROD1 6.04e-5 -1.1045 0.0 / Dake's formula values (field units)
/ 
```

---

**Keywords**

2479
WDRILPRI  Puts wells into the prioritized drilling queue

This keyword is used to put wells into the prioritized drilling queue and to define their drilling priority values and drilling unit numbers. The drilling priority values set here will override the values calculated using the drilling priority equation (see keyword DRILPRI).

The wells are opened in decreasing order of their drilling priority when they are needed to maintain a group rate target under group control by guide rate (GCONPROD, GCONINJE, GCONSALE) or when needed to maintain a group’s production potential (GDRILPOT). If a group (including the field) cannot meet its production target, the production well with the highest drilling priority that is subordinate to that group, and not subordinate to any other group under the same mode of production control, are opened automatically. If a group cannot meet its injection target, the injection well with the highest drilling priority that is subordinate to that group, and not subordinate to any other group under injection control for the same phase, will be opened automatically.

Well opening may be delayed by constraints on the rate of drilling new wells (see keyword WDRILTIM), the availability of drilling rigs (keyword GRUPRIG) and, in ECLIPSE 100, on the maximum number of open wells in any group (item 9 of keyword GECON). If a well cannot currently be drilled because of any of these constraints, wells with lower drilling priority in the queue are examined in decreasing priority order to see if there is another suitable well that can be drilled instead.

An alternative to the prioritized drilling queue is the sequential drilling queue, from which wells are opened in the sequence in which they are placed in the queue. Wells are placed in the sequential drilling queue using the keyword QDRILL. Only one type of drilling queue may exist at any given time in the run; the two types of queue cannot operate together.

The WDRILPRI keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

1. Well name, well name template, well list or well list template.
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Drilling priority value for the well(s).
   If the priority is zero the well is removed from the drilling queue.
   If the priority is greater than zero, the well’s drilling priority remains fixed at this value.
   If the priority is negative, the well’s drilling priority is calculated at specified intervals (see keyword DRILPRI) from its potential rates. For producers, it is calculated from the priority equation defined with keyword DRILPRI. For injectors the drilling priority is set equal to their injection potentials.
   DEFAULT: Negative
   Priority values do not have any units.

ECLIPSE 100 only
3. Drilling unit number

If set > 0 all wells with this particular drilling unit number is drilled at the same time. Also their drilling priorities (either calculated or user-specified) are added together when the drilling queue is scanned for the highest priority value, so that the most productive unit overall is selected if the well priorities reflect their production potentials. This facility allows, for example, wells in the same locality to be drilled consecutively once it has been decided to send a drilling rig there.

Once a unit has been selected for drilling, the drilling process for all its wells begins immediately. The drilling rig is logged as busy for the sum of the wells’ drilling times (keyword WDRILTIM item 2). However, all the wells begin flowing immediately unless they are flagged as closed while being drilled (WDRILTIM item 3). In that case each well is given a zero efficiency factor until its drilling process is finished, and each well is drilled consecutively in the order in which they were first declared with the WELSPECS keyword. If more than one rig is available, the drilling process is shared between them.

DEFAULT: 0

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Note: The drilling queue is not used for opening wells belonging to groups under prioritization group control (keyword GCONPRI), as the prioritization algorithm itself handles the opening and closing of these wells. Drilling priorities should not be confused with well priorities set with keywords PRIORITY and WELPRI.

See also keywords DRILPRI, WDRILRES, WELSOMIN, PRORDER and QDRILL.

Example

```
WDRILPRI
 'PROD*' -1.0 /
 PROD1 1000.0 /
 /
```

Keywords

2481
WDRILRES  Prevents two wells being drilled in the same block

After this keyword is used, any well in the drilling queue (keyword QDRILL or WDRILPRI) or on automatic opening (AUTO in keyword WCONPROD for example) will be prevented from opening if it is completed in the same grid block as a well that is already open. It is then removed from the drilling queue or automatic opening queue.

With this facility it is effectively possible to allocate a well to the first group that is able to support it, subject to its maximum allowed number of open wells set in keyword GECON. You must define a different well for each group, completed in the same location, and place them in the drilling queue or under automatic opening. These wells in fact all represent the same physical well. Thus when one well is opened the remaining wells must be left shut.

This keyword does not prevent wells being opened manually, for example, with keyword WELOPEN.

The keyword has no associated data.
Controls on the automatic drilling of new wells

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated at any item after item 1. The remaining items will assume their default values.

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template.

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

   If QDRILL is entered here, followed by the slash terminator, the option to prevent repeated premature opening of wells in the drilling queue will be invoked (see "Wells in the drilling queue").

2. Time taken to drill the well

   UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

   DEFAULT: 0.0

   This item limits the rate at which new wells can be opened either:

   • automatically (AUTO in keywords WCONPROD, WCONINJE, WCONINJ, WELOPEN),
   • or from the drilling queue (keyword QDRILL or WDRILPRI),
   • or on the closure of another well (item 9 in keyword WECON).

   A well with a non-zero drilling time requires a drilling rig. It will not be drilled if all the rigs available to the well are occupied up to the end of the timestep. When a drilling rig allocated to any superior group becomes free, the well’s drilling operation will occupy the rig for the specified time. Rig availability has no effect on wells being opened manually (for example with keywords WELOPEN or WCONPROD). Drilling rigs are allocated to groups with the keyword GRUPRIG.

3. Closure of well during drilling and workovers

   YES  The well is treated as shut during automatic drilling and workovers, by temporarily adjusting its efficiency factor (see keyword WEFAC) for the timesteps in which these processes occur. Its flows are then not included in the group totals and cumulative totals while it is being drilled or worked over. The closure time for workovers is set in keyword WORKLIM (it defaults to zero if WORKLIM is not entered). This does not affect wells being opened or recompleted ‘manually’.

   NO   The well starts flowing at the beginning of the timestep in which its drilling operation starts, and is not shut during workovers.

   Drilling is prevented if all the drilling rigs available to the well are occupied up to the end of the timestep. Otherwise the well will be opened at the beginning of the timestep.

   DEFAULT: NO

4. Compartment number for the well (a positive integer or zero).

   ECLIPSE 100 only
If set positive, all wells in that compartment will be temporarily shut while any well in the
compartment is drilled or worked over automatically. The temporary closure is performed by reducing
the wells’ efficiency factors for the timesteps in which an automatic drilling or workover occurs.
Wells being opened or recompleted ‘manually’ will have no effect.

A positive number may be supplied only if item 3 is set to YES.

DEFAULT: 0

(temporary closure is not performed in compartment number 0)

End the data record with a slash (/).

End the set of records with a blank record containing just a slash.

**Wells in the drilling queue**

If the controls in items 3 and 4 are applied to wells in the drilling queue, they may cause wells to be drilled
sooner than is really necessary. If a group cannot meet its flow target, a well from the drilling queue is
opened. But if the well is treated as shut during the drilling process, its full benefit is not realized until the
timestep after drilling has been completed. Its contribution during the timestep in which it is drilled may
therefore not be sufficient to enable the group to reach its target, and a second well from the drilling queue
may be opened (provided that there is still some free drilling rig time left in the timestep). For ECLIPSE
100, this effect is even more likely to occur if other wells in the same compartment are shut during the
drilling process. The loss of flow from these wells during the drilling process can be greater than the extra
contribution from the new well in the timestep in which it is drilled. Thus the immediate effect of drilling a
new well could be a net reduction in flow rate, and the drilling queue is scanned again if there is still some
free drilling rig time left in the timestep.

There is an optional switch to prevent the early drilling of additional wells in the drilling queue. It is
triggered by including an extra record in this keyword, having **QDRILL** in item 1 in place of a well name,
followed by the slash terminator:

```
QDRILL /
```

If any other item in this record is set to a non-default value, the record is treated as a standard one referring
to a well called **QDRILL**. If all other items in the **QDRILL** record are defaulted, the switch preventing
repeated drillings is set ON for the rest of the simulation (including any restarts). The switch prevents the
drilling queue being scanned during timesteps in which a drilling operation of non-zero duration is already
taking place (including drilling operations started in a previous timestep) on the same type of well
(producer and water injector for example) subordinate to the group requesting the drilling. Thus the
decision on whether to scan the drilling queue to open an additional well is taken only after any wells of the
same type and subordinate to the same group requesting the drilling are fully on stream.

If the opening of new wells from the drilling queue is one of the Group Production Rule actions specified in
keyword **PRORDER**, do not use the **QDRILL** switch in keyword **WDRILTIM**. Instead, it is better to set item
1 in the second record of keyword **PRORDER** to **NO**, as this restricts all the production rule actions and not
just drilling.

**Example**

```
WDRILTIM
-- well drilling close while compartment
-- name time drilling? number
'W1*'  30 /
'W2*'  25   YES  1 /
```

**Keywords**
ECONOMIC LIMIT DATA FOR PRODUCTION WELLS

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated at any item after item 1. The remaining items will assume their default values. Default values can be specified before the slash by null repeat counts of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template.
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Minimum oil production rate. User defined arguments can be set with keywords UDQ and UDADIMS.
   If the oil production rate falls below the specified minimum, the well is shut or stopped (see item 9 in keyword WELSPECS).
   However, if the well has any connections left on automatic opening (see item 6 of keyword COMPDAT) and which meet the minimum oil saturation requirements (if imposed with keyword WELSOMIN), one of these is opened instead, as soon as a workover rig is available.
   If item 10 is POTN, the minimum rate limit is applied to the well’s oil production potential instead of its actual rate.
   A zero or negative value switches off this limit.
   UNITS: m³/day (METRIC), stb/day (FIELD), scc/hr (LAB), m³/day (PVT–M)
   DEFAULT: 0.0

3. Minimum gas production rate. User defined arguments can be set with keywords UDQ and UDADIMS.
   If the gas production rate falls below the specified minimum, the well is shut or stopped (see item 9 in keyword WELSPECS).
   However, if the well has any connections left on automatic opening (see item 6 of keyword COMPDAT) and which meet the minimum oil saturation requirements (if imposed with keyword WELSOMIN), one of these is opened instead, as soon as a workover rig is available.
   If item 10 is POTN, the minimum rate limit is applied to the well’s gas production potential instead of its actual rate.
   UNITS: m³/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), m³/day (PVT–M)
   DEFAULT: 0.0

4. Maximum water cut. User defined arguments can be set with keywords UDQ and UDADIMS.
   The water cut is the ratio of water to total liquid (oil + water) surface volume production rates.

Keywords
A value of 0.0 switches off this limit.

UNITS: \( \text{sm}^3/\text{sm}^3 \) (METRIC), stb/stb (FIELD), scc/scc (LAB), \( \text{sm}^3/\text{sm}^3 \) (PVT-M)

DEFAULT: No limit

5. Maximum gas-oil ratio. User defined arguments can be set with keywords UDQ and UDADIMS.

A negative or zero value switches off this limit.

ECLIPSE 100

If a negative value is specified (- X), the well is not closed or worked over on violation of its water cut limit (item 4), or a group water cut limit (GECON) or a group water or liquid rate limit (GCONPROD), if the well has a gas-liquid ratio greater than the positive value X. This option is intended for wells that produce mainly gas and mainly oil at different stages of their life. The option allows the well to have a high water cut while it is a gas producer.

UNITS: \( \text{sm}^3/\text{sm}^3 \) (METRIC), Mscf/stb (FIELD), scc/scc (LAB), \( \text{sm}^3/\text{sm}^3 \) (PVT-M)

DEFAULT: No limit

6. Maximum water-gas ratio. User defined arguments can be set with keywords UDQ and UDADIMS.

A value of 0.0 switches off this limit.

UNITS: \( \text{sm}^3/\text{sm}^3 \) (METRIC), Mscf/stb (FIELD), scc/scc (LAB), \( \text{sm}^3/\text{sm}^3 \) (PVT-M)

DEFAULT: No limit

7. Workover procedure on exceeding water cut, GOR, WGR or GLR limit

NONE No action to be taken

CON Shut worst-offending connection

+CON Shut worst-offending connection and all those below it. (See keyword COMPORD) (In ECLIPSE 300, CON+ is interpreted as +CON for back-compatibility)

WELL Shut or stop the well (item 9 in keyword WELSPECS)

ECLIPSE 100

Plug back the well (see keyword WPLUG)

DEFAULT: NONE

8. End run flag

YES The run stops at the next report time if the well is shut or stopped for any reason after being opened

NO The run continues regardless

DEFAULT: NO

9. Name of well to be opened when this well is shut (the follow-on well)

If a well name is entered here, the named well is opened when the well specified in item 1 is closed automatically for any reason (for example on violating an economic limit). The named well must be fully specified as a SHUT producer or injector. Manual closure of the well in item 1 (for example using WCONPROD or WELOPEN) does not cause the named well to be opened.

Keywords

2487
This option can be used, for example, to open an injector when a producer closes after becoming uneconomic. If the injector has the same completions as the producer, the effect is equivalent to converting the producer into an injector, although in this case they is regarded as two separate wells.

If the well to be opened is given a non-zero drilling time in keyword WDRILTIM, the opening is subject to constraints on drilling rig availability (see keyword GRUPRIG).

DEFAULT: '  

10. Quantity to which the minimum economic limits in items 2, 3, 14 and 16 will be applied

RATE The limits are applied to the well’s actual flow rate. The minimum rate checks are bypassed for wells whose rate target is currently cut back under prioritization (GCONPRI), to prevent them being closed while temporarily flowing with a small target rate. For wells subordinate to groups under guide rate control (GCONPROD), the minimum rate limits will be applied unless the group has a zero production rate target.

POTN The limits are applied to the well’s production potential (that is the rate that the well could immediately achieve if all rate limits were removed, honoring only its BHP and THP limits, and any drawdown limit if requested in WELDRAW - see “Well potentials” in the ECLIPSE Technical Description). Thus if the well’s production rate was cut back to meet a group rate target, this does not trigger the minimum economic limit check. Wells under prioritization (GCONPRI) have their potentials checked against the minimum economic limits, even if they are not currently selected to flow, and are permanently closed if the potential is too small.

DEFAULT: RATE


If this value is set > 0.0 (and < 1.0), a secondary water cut limit is applied to the well. The application of this limit depends on the action selected in item 12. The application associated with the actions LAST and RED are described in item 12. For all other actions, the secondary limit is applied as described below.

If the well has been subjected to a number of workovers due to the violation of the primary water cut limit (item 4), it is shut in when its last remaining connection is closed in a workover. However, if the well has a secondary water cut limit set, it will be reopened at the beginning of the next timestep and the secondary water cut limit will replace the primary water cut limit in the economic limit tests. Also, all its connections that were closed in the previous workovers are reopened. The connections may be closed again in subsequent workovers whenever the well exceeds the new water cut limit, depending on the workover action chosen in item 12.

Thus, the well can be kept within its primary water cut limit for as long as possible by closing its worst offending connections. But when its last connection exceeds the primary water cut limit, the whole well is given a second life for as long as it able to produce within the secondary water cut limit. Obviously, the secondary water cut limit should be set to a value significantly greater than the primary limit in item 4.

The workover procedure in item 7 should be set to CON, +CON or PLUG.

UNITS: sm$^3$/sm$^3$ (METRIC), stb/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)

DEFAULT: 0.0 (No secondary water cut limit)

12. Workover action on exceeding the secondary water cut limit defined in item 11

NONE  Do nothing
CON  Shut worst-offending connection

+CON  Shut worst-offending connection and all those below it (see keyword COMPORD)

WELL  Shut or stop the well (item 9 in keyword WELSPECS)

ECLIPSE 100

PLUG  Plug back the well (see keyword WPLUG)

LAST  The secondary water-cut is applied only to the last open connection in the well. (Connections on automatic opening class as open in this context.) This action differs from the ones above in that, when the well has only one connection left open and it begins to exceed its primary water-cut limit again, the well remains open until it exceeds its secondary water-cut limit but the other connections closed in previous workovers will not be reopened.

ECLIPSE 100

RED  The well is worked over each time it violates its primary water-cut limit, as normal. But each workover attempts to close sufficient connections to reduce the well’s water-cut to below the specified secondary limit. Clearly, in this case, the secondary value should be less than the primary value. Assuming the well can be worked over to achieve this secondary value, it is not worked over again until the primary value is again violated. This option has the effect of requiring fewer workovers but closing more connections per workover.

DEFAULT: The same action as defined in item 7

13. Maximum gas-liquid ratio

The workover procedure on exceeding this limit is defined in item 7.

A value of 0.0 switches off this limit.

UNITS: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)

DEFAULT: No limit

14. Minimum liquid production rate

If the liquid production rate falls below the specified minimum, the well is shut or stopped (see item 9 in keyword WELSPECS).

If item 10 is POTN, the minimum rate limit is applied to the well’s liquid production potential instead of its actual rate.

A zero or negative value switches off this limit.

UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: 0.0

ECLIPSE 300 only

15. Maximum temperature

(available only with the Thermal option; keyword THERMAL in the RUNSPEC section.)

The well is shut if this temperature is exceeded.

UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

16. Minimum reservoir fluid production rate

If the reservoir fluid production rate falls below the specified minimum, the well is shut or stopped (see item 9 in keyword WELSPECS).
If item 10 is POTN, the minimum rate limit is applied to the well’s reservoir fluid production potential instead of its actual rate.

A zero or negative value switches off this limit.

UNITs: rm³/day (METRIC), rb/day (FIELD), rcc/hr (LAB), rm³/day (PVT-M)

DEFAULT: 0.0

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also keywords COMPLUMP, WLIMTOL, WORKLIN, GRUPRIG and PRORDER.

**Example**

```
WECON
  PROD1  500 1*  0.8  10.0  1* CON N INJ1 /
  PROD2  100  5*                 Y /
  PROD3  100 /
/```

*Keywords*

2490
ECONCMF Economic limit data on component mole fractions for production wells

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated at any item after item 1. The remaining items will assume their default values. Default values can be specified before the slash by null repeat counts of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template.
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Component index
   An integer between 1 and the value set in keyword COMPS.

3. Maximum mole fraction of the component specified by item 2 in the produced wellstream. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.
   A value of 1.0 switches off this limit.

4. Workover procedure on exceeding the mole fraction limit
   NONE  No action to be taken
   CON   Shut worst-offending connection
   +CON  Shut worst-offending connection and all those below it (see keyword COMPORD)
   WELL  Shut or stop the well (item 9 in keyword WELSPECS)

   DEFAULT: NONE

5. End run flag
   YES   The run stops at the next report time if the well is shut or stopped for any reason after being opened
   NO    The run continues regardless

   DEFAULT: NO

See also keywords COMPLUMP and WLIMTOL

Example

```
ECONCMF
  PROD1 1 0.05 +CON / 
/ 
```

Keywords
ECLIPSE Reference Manual

WECONINJ Economic limit data for injection wells

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated at any item after item 1. The remaining items assume their default values. Default values can be specified before the slash by null repeat counts of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

   If the injection rate falls below the specified minimum, the well is shut or stopped (see item 9 in keyword WELSPECS). This rate applies to the phase specified in item 2 of the keyword WCONINJ or WCONINJE.
   If item 3 below is POTN, the minimum rate limit is applied to the well’s oil/water/gas injection potential instead of its actual rate.
   A zero or negative value switches off this limit.
   UNITS: As appropriate for item 2 of WCONINJ/WCONINJE
   DEFAULT: 0.0

3. Quantity to which the minimum economic limits in item 2 will be applied.
   RATE The limits apply to the well’s actual injection rate. For wells under group control (GCONINJE), the minimum rate limit will be applied unless the group has a zero injection rate target.
   POTN The limit applies to the well’s injection potential (that is the rate that the well could immediately achieve if all rate limits were removed, honoring only its BHP and THP limits - see "Well potentials" in the ECLIPSE Technical Description). Thus if the well’s injection rate was cut back to meet a group rate target, this will not trigger the minimum economic limit check.
   DEFAULT: RATE

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also keyword WLIMITOL.

Example

<table>
<thead>
<tr>
<th>WECONINJ</th>
</tr>
</thead>
<tbody>
<tr>
<td>INJ1 500 RATE /</td>
</tr>
</tbody>
</table>

Keywords

2492
INJ2 800 POTN /
**Tracer economic limits for production wells**

**WECONT** is used to set tracer economic limits for production wells, including polymer and brine.

**WECONT** is followed by any number of records, described below, each record terminated by a slash (/). A record may be terminated early at any item; the remaining items assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted. There are two types of record; one describes the connection and the second lists the tracers and their limits.

The set of records must end with a blank record, containing only a slash (/).

**Records for the connection**

Records of the first kind can contain some or all of the following items:

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Workover procedure when a limit is violated

   **NONE**  No action to be taken

   **CON**   Shut worst-offending connection

   **+CON**  Shut worst-offending connection and all those below it (see keyword COMPORD)

   **WELL**  Shut or stop the well (item 9 in keyword WELSPECS)

   **PLUG**  Plug back the well (see keyword WPLUG)

   **DEFAULT:** NONE

3. End run flag

   **YES**  The run stops at the next report time if the well is shut or stopped for any reason after being opened

   **NO**   The run continues regardless

   **DEFAULT:** NO

4. Name of well to be opened when this well is shut (the follow-on well)

   If a well name is entered here, the named well is opened when the well specified in item 1 is closed automatically for any reason (for example on violating an economic limit). The named well must be fully specified as a SHUT producer or injector. Manual closure of the well in item 1 (for example using WCONPROD or WELOPEN) does not cause the named well to be opened.

   This option can be used, for example, to open an injector when a producer closes after becoming uneconomic. If the injector has the same completions as the producer, the effect is equivalent to converting the producer into an injector, although in this case they are regarded as two separate wells.
If the well to be opened is given a non-zero drilling time in keyword `WDRILTIM`, the opening is subject to constraints on drilling rig availability (see keyword `GRUPRIG`).

DEFAULT: ‘ ’ (no follow-on well)

End the record with a slash (/).

**Records for the tracers**

A maximum of three subsequent records can follow, for each well. These records have some or all of the following items.

1. Tracer name (Use PLY for a polymer and BRI for brine)
2. Maximum total tracer rate
   DEFAULT: No limit
3. Maximum total tracer concentration (not available for partitioned tracers)
   DEFAULT: No limit
   The following are only available for two-phase tracers
4. Maximum free tracer rate
   DEFAULT: No limit
5. Maximum free tracer concentration
   DEFAULT: No limit
6. Maximum solution tracer rate
   DEFAULT: No limit
7. Maximum solution tracer concentration
   DEFAULT: No limit.

End the record with a slash (/).

End the data for this well with a blank line containing just a slash(/).

Note that this means that the keyword will end with two blank lines containing just slashes.

**Notes**

- No units are given for the tracer rates. The rates are those of the carrier flow.
- Since the tracers are solved after the main reservoir simulation if it is subsequently discovered that a limit is violated, the timestep is always repeated with the action being enforced from the start of the timestep. Even with this it may be that the ‘action’ lags the violation by a timestep.
- If the limit is on tracer concentration it is possible for the ‘action’ to lag the apparent violation by 2 timesteps. This is because the concentration is a derived quantity, and in deriving the value at the well ECLIPSE uses the latest available information. This may result in a value (slightly) different from the one calculated, and reported later, from the tracer calculation. If this is problematic it may be worth considering using ACTION keywords to replace the keyword entirely or to use ACTION keywords to reduce the timestep in the run-up to the limit being violated.
- This keyword needs to end with two lines containing only slashes. See the examples.
Examples

Example 1

```
WECONT
   PROD1 CON   /
   TR1 1000.0 0.5   /
   TR2 2* 1* 0.8 2*   /
   TR3 1500.0 0.1   /
   PROD2 CON+   /
   TR3 100.0 0.05   /
   PROD3 WELL/   
   TR4 200 5*   /
   PLY 1* 0.7   /
   /
```

Example 2

```
WECONT
   PROD4 CON   /
   TR5 1000.0 0.5   /
   TR1 2* 1* 0.8 2*   /
   TR3 1500.0 0.1   /
   /
```

Example 3

```
WECONT
   PRODUCER CON+   /
   BRI 1000.0 0.1   /
   /
```
Sets well efficiency factors (for downtime)

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Efficiency factor for the well(s)

This is the fractional time for which the well is operational. For example, if the well is 'down' for 10 percent of the time, its efficiency factor will be 0.9.

The efficiency factor must be greater than zero.

DEFAULT: 1.0

3. **Extended Network option only** Should a well's efficiency factor be included when calculating the branch flows and pressure losses in the extended network option? (See “Well and group efficiency factors” in the ECLIPSE Technical Description).

- **YES** The extended network branch pressure losses is calculated using the well's time-averaged flow rate (that is the well rate multiplied by its efficiency factor).

- **NO** The extended network branch pressure losses is calculated using the well's maximum flow (that is the well rate not multiplied by its efficiency factor).

This flag applies only to the extended network model (constructed with keywords BRANPROP and NODEPROP). If the Standard Network Model is being run (GRUPNET, GNETINJE), the time-averaged well flows are always used to calculate the branch pressure losses.

DEFAULT: **YES**

**CAUTION:** Do not select the NO option in item 3 if group rate constraints are to be handled by automatic chokes (if NODEPROP item 3 is YES for any network node). The constraints are then applied to the flow in the network (the sum of the wells' maximum flow rates), which are inconsistent with the group flows (which are always the sum of the subordinate wells' time averaged flows).

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

The efficiency factors for wells not specified with this keyword are defaulted to 1.0.

**Treatment of downtime**

Each well is solved at its full flowing conditions, and the well rates and pressures given in the well reports describe the state of the flowing well. However, the flow rates are multiplied by the well’s efficiency factor when:

- The well rates are summed to give the group flow rates,
• The well rates are multiplied by the timestep and added to the cumulative flows,
• The connection flows are applied as source/sink terms in the reservoir material balance equations.

This essentially treats the wells as going down individually for random short intervals within each timestep, rather than all going down simultaneously. (If all the wells in a group are brought down together for regular maintenance, then the efficiency factor should be applied to the group with keyword GEFAC, currently available in ECLIPSE 100 only.) The group and field flows represent the average flow rates over the period. For example, if the efficiency factors are all 0.5, the well’s individual flow rates sum to twice the group rate, but only half of them are flowing at any given instant on average.

If a group flow target is applied (for example with GCONPROD), this is interpreted as the target average flow rate of the group with the wells flowing intermittently according to their efficiency factors. The effect of the wells’ downtime is to make the wells flow proportionately more while they are operating. For example, consider a group with a target oil rate of 10,000 stb/day to be divided equally between its 10 wells. If all wells had zero downtime, each would produce 1,000 stb/day. However, if they all have an efficiency factor of 0.8, each well must produce 1,250 stb/day while it is flowing to make up the group’s target rate, provided that the wells’ flowing conditions at this rate do not violate any of their operating limits.

If wells are receiving gas lift, their lift gas injection rates are multiplied by their efficiency factors when summing the group lift gas supply rates. The group lift gas supply rates are needed if there are restrictions on the total lift gas supply (that is, if using the GLIFTLIM keyword or the ECLIPSE 100 gas lift optimization facility).

When the standard network option is in use, the network pressure losses are calculated at the group average flow rates (that is the sum of the well rates multiplied by their efficiency factors). However, the extended network model, in which the network structure can be different from the group hierarchy, offers more flexibility (see “Well and group efficiency factors” in the ECLIPSE Technical Description). If WEFAC keyword item 3 is set to NO for each well, the pressure losses in the extended network are calculated using the maximum flows instead (that is the sum of the well rates not multiplied by their efficiency factors). The branch flows are then of course be different from the reported flows of the corresponding groups, which are their time-averaged rates.

An efficiency factor will still be applied to a well whose status is currently set to STOP. This includes flow across the well bore if the crossflow model is active. As the well efficiency factor assumes that the well is solved at operating conditions and that there is zero flow across the well during downtime, WEFAC is inappropriate for modeling downtime of wells with crossflow. To model crossflow accurately during downtime it is advised that the well’s status is defined explicitly by manually opening and shutting the well for the appropriate time rather than using WEFAC.

**Example**

```

table

WEFAC
  'P*'  0.9  /
  PROD2  0.8  /
/
```
WELCNTL  Resets a well control mode and operating target

This keyword can be used to reset the control mode and target value for a well, without having to re-specify all the other quantities required by the control keywords WCONPROD or WCONINJE. These other quantities are left unchanged, including the open/shut status of the well. The well control data must initially have been fully specified using WCONPROD, WCONINJE, WCONHIST or WCONINJH.

This keyword is similar to the WELTARG keyword, but it also allows the control mode to be changed, not just an operating target or limit. Changing the control mode in this way, particularly when using a defaulted item 3, can be useful when switching from history matching mode to predictive mode during a simulation.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Definition of the new control mode for the well

   ORAT  Oil rate
   WRAT  Water rate
   GRAT  Gas rate
   LRAT  Liquid rate
   CRAT  Linearly combined rate (coefficients must be set with LINCOM)
   RESV  Reservoir fluid volume rate
   BHP   Bottom hole pressure
   THP   Tubing head pressure (A non-zero VFP table number must first have been set)

   ECLIPSE 300 only  WGRA  Wet gas rate
   ECLIPSE 300 only  NGL   NGL rate
   ECLIPSE 300 only  CVAL  Calorific production rate
   ECLIPSE 300 only  REIN  Reinjection fraction
   ECLIPSE 300 only  STRA  Steam rate (Available only with the Thermal option)
   ECLIPSE 300 only  SATP  Saturation pressure offset (Available only with the Thermal option)
   ECLIPSE 300 only  SATT  Saturation temperature offset (Available only with the Thermal option)

3. New value of the operating target for this control mode.
UNITS: The units depend upon the quantity chosen. The appropriate units are listed in the specification for keywords \texttt{WCONPROD} and \texttt{WCONINJE}.

DEFAULT: A defaulted value results in ECLIPSE using the corresponding flowing value for the well at the end of the previous timestep as the target for the control in item 2.

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also keyword \texttt{WELTARG}.

\textit{Note:} One of the primary purposes of this keyword is to make the transition from history matching mode to prediction mode easier to specify. However, if this keyword is used to change the control mode of a well which was previously specified using \texttt{WCONHIST} or \texttt{WCONINJH}, then the old historical \textbf{target} (for example, oil rate) will still be present as a \textbf{limit}. This limit may interfere with the new control mode specified with the \texttt{WELCNTL} keyword, and cause the well to switch back to the old control mode. In order to avoid this, remove the old limit by using the \texttt{WELTARG} keyword to set this limit to a large value (1.0E+8 for example).

Example

\begin{verbatim}
WELCNTL
  PROD1  ORAT  1500 /
  PROD2  BHP  /
/
\end{verbatim}
WELDEBUG  Controls on debug output for individual wells

This keyword is used control debug output for individual wells.

**Note:** This output is primarily intended as an aid to program development and debugging, and it should not be requested for normal program runs.

The keyword is followed by any number of lines, each containing the following data and each terminated with a slash (/). The last line must be followed by a line containing only a slash.

1. **Well name, well name template, well list or well list template**
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword **WLIST**.

2. **Debug output control.**

   **ECLIPSE 100**  
   This is a 4-figure integer:
   - The 1\textsuperscript{st} figure controls debug from well flow and property calculations.
   - The 2\textsuperscript{nd} figure controls debug from connection flow calculations.
   - The 3\textsuperscript{rd} figure controls debug from calculations involving VFP tables.
   - The 4\textsuperscript{th} figure controls debug from well terms in the solver.

   **ECLIPSE 300**  
   This should be either 0, 1 or 2. If a 4-figure integer is read, only the first figure is regarded as significant.
   - A value of 1 or more turns the relevant debug output switch on. Higher integers result in greater quantities of debug output. A value of zero turns the debug output off.

   **ECLIPSE 300**  
   This keyword is ignored if **DEBUG3** (11) is set to either 1 or 2, in which case debug is output for all wells.

**Example**

```
WELDEBUG
    PROD1  2120 /
    PROD2  1001 /
```

Keywords

2501
WELDRAW  Sets maximum allowable drawdown for production wells

This keyword can be used to set the maximum drawdown allowed for each production well. At each timestep, the maximum drawdown $D_{\text{max}}$ is converted into a maximum production rate $Q_{\text{max}}$ of a selected phase, either liquid (oil + water) or gas, using the relationship

$$Q_{\text{max}} = D_{\text{max}} \sum_j [T_{wj} M_j]$$  \hspace{1cm} \text{Eq. 3.226}

where the term

$$\sum_j [T_{wj} M_j]$$

denotes the product of the connection transmissibility factor $T_w$ and the phase’s mobility $M$ in the connection, summed over all open connections in the well.

This relationship effectively defines the drawdown as

$$D = P_{av} - P_w$$  \hspace{1cm} \text{Eq. 3.227}

where

$P_w$ is the well’s bottom hole pressure, and

$P_{av}$ is a PI-weighted average of the pressures $P$ in the grid blocks containing the well's open connections, given by

$$P_{av} = \frac{\sum_j [T_{wj} M_j (P_j - H_{wj})]}{\sum_j [T_{wj} M_{wj}]}$$  \hspace{1cm} \text{Eq. 3.228}

$H_{wj}$ is the hydrostatic wellbore pressure head between the connection $j$ and the well’s bottom hole pressure reference depth.

For a more detailed definition of these terms, see "Well inflow performance" in the ECLIPSE Technical Description.

As an alternative to limiting the PI-weighted average of the drawdowns within the grid blocks containing the well’s open connections, it is also possible to apply the limit to the maximum drawdown within the set of grid blocks containing open connections to the well. This option would be particularly useful for wells completed in two or more layers with poor vertical communication and disparate pressures. The choice is determined by item 5 of the keyword. If this item is set to MAX, ECLIPSE calculates the minimum allowed BHP as

$$P_{w,\text{min}} = \max_j (P_j - H_{wj} - D_{\text{max}})$$  \hspace{1cm} \text{Eq. 3.229}

which it converts into a maximum production rate using the current fluid mobilities,

$$Q_{\text{max}} = \sum_j [T_{wj} M_j (P_j - H_{wj} - P_{w,\text{min}})]$$  \hspace{1cm} \text{Eq. 3.230}

When a well’s production rate is being constrained by its drawdown limit, the well will be on liquid or gas rate control (depending on the phase selected), producing at the rate of $Q_{\text{max}}$ given above. If you have set a
specific rate limit for the selected phase (using keyword \texttt{WCONPROD} or \texttt{WELTARG}), the flow rate is the minimum of $Q_{\text{max}}$ and the user-specified limit.

Since $Q_{\text{max}}$ depends on the mobility of the selected phase, which can change at each Newton iteration of a timestep, a well constrained by its drawdown limit could have a different flow rate target at each iteration. This may slow down the convergence rate of the Newton iterations. Therefore $Q_{\text{max}}$ is updated only during the first \texttt{NUPCOL} iterations of each timestep, just like the target rates of wells under group control (see keyword \texttt{NUPCOL}). For any remaining iterations, $Q_{\text{max}}$ is left unchanged, which may cause the actual drawdown to drift slightly away from the limit. Normally this drift will be negligible provided that the phase’s mobility does not change too much during a single timestep. Obviously the selected phase should be the dominant one for the well, that is, liquid for oil wells and gas for gas wells.

Drawdown control can be used in combination with group control and other well constraints.

The keyword \texttt{WELDRAW} is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the \textit{ECLIPSE Technical Description} for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword \texttt{WLIST}.

2. Maximum allowable drawdown for the well(s). This is a user defined argument that can be set with keywords \texttt{UDQ} and \texttt{UDADIMS}.

\textbf{UNITS}: bars (\texttt{METRIC}), psi (\texttt{FIELD}), atm (\texttt{LAB}), atm (\texttt{PVT-M})

3. Phase selected for rate limit.

\texttt{LIQ} For oil wells (unless fully gassed out)

\texttt{GAS} For gas wells (unless fully watered out)

At each timestep, the drawdown limit will be converted into a maximum production rate limit for this phase, using the current value of the phase’s mobility in each connection.

\textbf{DEFAULT}: \texttt{GAS} if the well’s preferred phase (\texttt{WELSPECS} item 6) is gas, \texttt{LIQ} in all other cases.

4. Do you wish the drawdown limit to be taken into account when calculating the well’s production potential?

\texttt{YES} The well’s production potential depends on its BHP, THP and drawdown limits

\texttt{NO} The well’s production potential depends only on its BHP and THP limits.

Production potentials may be output in the well reports (\texttt{WELSPECS} item 7) and the \texttt{SUMMARY} file (\texttt{WOPP} for example). ECLIPSE uses them to calculate well guide rates or priorities for group control. See "Well potentials" in the \textit{ECLIPSE Technical Description}.

\textbf{DEFAULT}: \texttt{NO}

5. Do you wish to limit the PI-weighted average of the drawdowns within the grid blocks containing the well’s connections, or the maximum drawdown within the set of connection grid blocks?
AVG  Limit the PI-weighted average drawdown, according to equations 3.226 to 3.228

MAX  Limit the maximum drawdown, according to equations 3.229 and 3.230

DEFAULT: AVG

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

**Example**

```
WELDRAW
PROD1 300 LIQ YES MAX /
PROD2 200 /
GPROD 200 GAS 1* MAX /
/
```
Sets summary mnemonic WPWEM for individual wells

This keyword is used to set the summary keyword WPWEM for individual wells. The value is reset to zero at the end of the timestep.

The keyword is followed by any number of lines, each containing the following data and each terminated with a slash (/). The last line must be followed by a line containing only a slash.

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Integer value to be output in the summary keyword WPWEM for this well or group of wells. This value is reset to zero after the timestep.

Example

```
WELEVNT
  PROD1 20 /
  PROD2 100 /
/
```
**WELLCF**

**Specifies behavior of wells with conductive fracture**

The WELLCF keyword should be used to modify the default behavior of wells intersected by conductive fractures modeled using the single medium conductive fracture formulation (see "Dual porosity" in the ECLIPSE Technical Description for further information). The **SCFDIMS** keyword must be present in the RUNSPEC section.

The keyword is followed by a number of records, each record describing the behavior required when a particular well is intersected by one or more fractures. The record fields are:

1. **Name** Name of the well concerned
2. **CUT, NOCUT** Whether or not the well is intersected by fractures
3. **CCF_MULT** CCF Multiplier to apply (whether CUT or NOCUT)
4. **IC** I-coordinate of the connection to apply the multiplier to
5. **JC** J-coordinate of the connection to apply the multiplier to
6. **KC** K-coordinate of the connection to apply the multiplier to
7. **IC1** Completion start index to apply multiplier to
8. **IC2** Completion end index to apply multiplier to

Each record should be terminated by a slash (/), and the total keyword data terminated by one final slash.

The **CUT / NOCUT** item must be entered; it has no default. If **NOCUT** is specified, then the saturation table index for that block will revert to that originally specified, and the well CCF modified so as to remove most of the effect of the single medium conductive fractures feature.

The CCF multiplier given is applied to the well completions specified, regardless of the value of the **CUT / NOCUT** item. It has a default of unity.

The well block specifications may be given as either single block entries {I,J,K}, one per line of data entry corresponding to a particular connection, or as a completion index range (IC1-IC2). Both methods may be defaulted with zero values, in which case all connected blocks are affected by the **CUT / NOCUT** item and the **CCF_MULT** given. If both methods are employed at the same time, then the connections chosen will satisfy both criteria.

**Note:** The WELLCF keyword behaves similarly to the **WPIMULT** keyword in that if a well completion is specified again, using **COMPDAT**, then in order to keep the required CCF multiplier, the WELLCF keyword and data must be re-specified. Also, multiple WELLCF keywords will result in cumulative multipliers on the CCF.

**Example**

```
WELLCF
 'P1' CUT 1.0 5*/
```

---

**Keywords**

2506
WELLCOMP \ Specifications a well completion

Specifies the position and properties of one or more well completions. This must be entered after the WELLSPEC keyword defining the appropriate well.

This keyword is retained for back-compatibility; you are encouraged to use keyword COMPDAT instead, which is compatible with both simulators.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form $n^*$, where $n$ is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. The I-index of the completion. If set to 0, or defaulted, this takes the value specified in WELLSPEC.

3. The J-index of the completion. If set to 0, or defaulted, this takes the value specified in WELLSPEC.

4. The smaller K-index of the completions
   DEFAULT: 1

5. The higher K-index of the completions
   DEFAULT: 1

6. The saturation table for well mobilities.
   If set to 0, or defaulted, the cell saturation table number is used - that is, the value specified with SATNUM.

7. The well bore diameter.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   DEFAULT: 1 ft = 0.3048 m

8. The completion connection factor.
   If this is not entered a default value is calculated using the Peaceman formula discussed in "The connection transmissibility factor" in the ECLIPSE Technical Description.

   If a value > 0.0 is entered, the completion connection factor is set to this value. The value will also be used to calculate either the connection's effective Kh or the denominator term $\ln(r_o/r_w) + S$ according to the following rules:

   - If the Kh value in item 11 is defaulted or set negative, the Kh value is calculated from the connection transmissibility factor and the denominator term. The denominator term is calculated from the grid block dimensions, as described in the "Well inflow performance" in the ECLIPSE Technical Description.
If the Kh value is set > 0.0, the denominator term is calculated from the connection transmissibility factor and the given Kh value, instead of the well bore radius and the grid block dimensions.

If the Kh value is set = 0.0, it is replaced by the effective Kh of the grid block, then used to calculate the denominator term in the same way as if it had been set > 0.0.

The denominator term is used when calculating the well’s productivity/injectivity index and when calculating the effects of a D-factor (see "Well inflow performance" in the ECLIPSE Technical Description). Thus, when the connection factor is set to a positive value, the results of these calculations depend on the setting of the Kh value in item 11.

Note: If a positive value is entered for the connection transmissibility factor, this should include the effect of any skin entered in item 11, but exclude the effect of any D-factor. The effect of a D-factor is modeled separately by adjusting the mobility (see "The flow-dependent skin factor" in the ECLIPSE Technical Description).

**UNITS:** cP.rm\(^3\)/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm\(^3\)/day/atm (PVT-M)

9. The skin factor
   DEFAULT: 0

10. The saturation table for imbibition (when Hysteresis option used).
    If set to 0, or defaulted, the cell imbibition table number will be used. This is the value specified with IMBNUM.

11. Well Kh (permeability x thickness) value.
    If defaulted, values of \(K_h\) are taken from cells in which the well is completed.
    
    If a positive value for the connection transmissibility factor is entered in item 8, the calculation of the denominator term depends on whether the Kh value is set or defaulted. See item 8.
    
    This quantity is required for calculating the connection transmissibility factor (if a default or zero value was supplied for item 8). It is ignored if not required.
    
    **UNITS:** mD.m (METRIC), mD.ft (FIELD), mD.cm (LAB), mD.m (PVT-M)
    
    DEFAULT: Negative

12. Direction in which the well penetrates the grid block.
    This should be one of:
    
    X The well penetrates the grid block in the X direction
    Y The well penetrates the grid block in the Y direction
    Z The well penetrates the grid block in the Z direction
    FX The well is assumed completed in a fracture in the x-direction
    FY The well is assumed completed in a fracture in the y-direction
This item is used in calculating the transmissibility factor of the connection if it was not supplied in item 8. In Cartesian grids the Peaceman formula is used, taking the components of permeability and grid block dimensions appropriate to the direction of penetration (see "The connection transmissibility factor" in the ECLIPSE Technical Description). Note that the formula assumes that the well completely penetrates a rectangular block through its center in a direction perpendicular to two of its faces.

In radial grids, only penetration in the Z direction is allowed.

If a value other than Z is specified, HWELLS must be included in the RUNSPEC section If FX or FY is specified, FWELLS must be included in the RUNSPEC section

DEFAULT: Z

See also "Hysteresis" in the ECLIPSE Technical Description

Examples

Example 1
Well P11 is completed in cells (1,1,3) and (1,1,4), with a diameter of 0.5 and well I5 in cell (7,7,1), with a diameter of 0.3ft. Both use the first saturation function table.

```
WELLCOMP
P11 1 1 3 4 1 0.5 /
I5 7 7 1 1 1 0.3 /
```

Example 2
In some cases it may be necessary to assign data values to each completion of a well individually, in which case a separate line of data should be entered for each line. Suppose that from flow meter analysis the Kh values of layers 3, 4 and 5 are 73, 94 and 35 respectively. The well is completed vertically in location (8,8), with a 0.5 ft diameter.

```
WELLCOMP
PRI 8 8 3 3 1* 0.5 3* 73 / --Layer 3
PRI 8 8 4 4 1* 0.5 3* 94 / --Layer 4
PRI 8 8 5 5 1* 0.5 3* 35 / --Layer 5
```

Example 3
The completions of a given well need not share the same I,J values, so that a well may be deviated out of a column. For example, a well completed in layers 1 to 4 of column (5,18) and layers 7 to 10 of column (6,18) might have:

```
WELLCOMP
WW1 5 18 1  4 1* 0.4 /
WW1 6 18 7 10 1* 0.5 /
```

Keywords

2509
WELLDIMS Well dimension data

The data consists of some or all of the following items, describing the dimensions of the well data to be used in the run. The data must be terminated by a slash (/).

1. The maximum number of wells in the model.
   - DEFAULT in ECLIPSE 100: 0
   - DEFAULT in ECLIPSE 300: 10

2. The maximum number of connections per well (that is the maximum number of grid blocks connected to any one well).
   - DEFAULT in ECLIPSE 100: 0
   - DEFAULT in ECLIPSE 300: Number of layers in field

3. The maximum number of groups in the model.
   - DEFAULT in ECLIPSE 100: 0
   - DEFAULT in ECLIPSE 300: 5

4. The maximum number of wells in any one group (or child groups in a group; keyword GRUPTREE). This is also the maximum number of wells in a particular local grid refinement (keyword WELSPECL).
   - ECLIPSE 100 only: DEFAULT: 0
   - ECLIPSE 300 only: DEFAULT: 10

5. The maximum number of stages per separator, including the last stage, which represents the stock tank (see keywords SEPCOND and FIELDSEP).
   - DEFAULT: 5

6. The maximum number of well streams (see keyword WELLSTRE).
   - DEFAULT: 10

7. The maximum number of mixtures (see keywords WINJMIX and WINJORD).
   - DEFAULT: 5

8. The maximum number of separators, including the field separator (see keywords SEPCOND and FIELDSEP).
   - DEFAULT: 4

9. The maximum number of mixture items in a mixture (see keywords WINJMIX and WINJORD).
   - DEFAULT: 3

10. The maximum number of completions allowed to cross domain boundaries within a parallel run (see "Parallel option" in the ECLIPSE Technical Description).
    - DEFAULT: 0

Keywords

<table>
<thead>
<tr>
<th>x</th>
<th>ECLIPSE 100</th>
<th>ECLIPSE 300</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SPECIAL</td>
<td>RUNSPEC</td>
</tr>
</tbody>
</table>

| x | GRID | EDIT | PROPS | REGIONS | SOLUTION | SUMMARY | SCHEDULE |

2510
11. The maximum number of well lists that a well may belong to at any one time (see keywords \texttt{WLIST}
and (ECLIPSE 300 only) \texttt{WLISTDYN}).
   \begin{itemize}
   \item \textit{DEFAULT:} 1
   \end{itemize}

\textit{ECLIPSE 300 only} 12. The maximum number of dynamic well lists in the simulation (see keyword \texttt{WLISTDYN}).
   \begin{itemize}
   \item \textit{DEFAULT:} 1
   \end{itemize}

\textit{ECLIPSE 300 only} 13. The maximum number of secondary wells in the simulation.
   \begin{itemize}
   \item \textit{DEFAULT:} 10
   \end{itemize}

\textbf{Example}

```
WELLDIMS
  20 4 4 5 10 2 4 /
```
WELLGR  

**Specifies well guide rates**

This keyword specifies the user guide rate for a well.

This keyword is retained for back-compatibility; you are encouraged to use keyword `WGRUPCON` instead, which is compatible with both simulators.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Well name, well name template, well list or well list template.

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the *ECLIPSE Technical Description* for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword `WLIST`.

2. A character specifying whether the well is available for group control.
   - Y Well is available for control by a parent group
   - N Well is not available for control by a parent group

   DEFAULT: Y

3. The guide rate value.

4. The type of guide rate being specified.
   This must be one of the following:

   - **OIL**  Oil production guide rate
   - **WAT**  Water production guide rate
   - **GAS**  Gas production guide rate
   - **LIQ**  Liquid production guide rate
   - **COM**  Linearly combined production guide rate (coefficients must be set with `LINCOM`)
   - **WG**  Wet gas production guide rate
   - **VP**  Voidage production guide rate
   - **IG**  Gas surface volume injection guide rate
   - **IW**  Water surface volume injection guide rate
   - **VG**  Gas voidage injection guide rate
WV  Water voidage injection guide rate
VI  Injection voidage guide rate (sets VG and VW)
RV  Reservoir voidage guide rate (sets VP, VG and VW)
INJ Injection guide rate (sets IG and IW)
NONE No guide rate specified, use potentials
ALL  Set all guide rates

Only the first two characters are significant.

By default, guide rates are set to the potential production or injection values of a group. These are normalized when used to distribute a group target over the sons of a group. In general, it is best to either specify all the guide rates for the sons of a group, (to specify the required distribution of targets explicitly), or to allow the default potential guide rates to be used.

It is usual to set one production guide rate (one of OI, WA, GA, LI, WG, VP), one water injection guide rate (one of IG, VG) and one gas injection guide rate (one of IW, VW). The others in each of these three classes are set using flow ratios from the value specified. The ALL option, VI, RV and INJ options allow several guide to be set explicitly.

Example

```
WELLGR
245 Y 100.0 INJ /
```

See also

- "Group production control options" in the ECLIPSE Technical Description
- "Group injection control options" in the ECLIPSE Technical Description
**WELLINJE**

**Specifies well injection targets**

This keyword specifies the injection control data for a well. Both the nature of the fluid to be injected, and the target and limiting rates at which injection is to occur are defined.

This keyword is retained for back-compatibility; you are encouraged to use keyword `WCONINJE` instead, which is compatible with both simulators, unless you wish to use the Thermal Option items which are not available with `WCONINJE`.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash (/), the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword `WLIST`.

2. A character string specifying the fluid to be injected.

   In black oil mode this may only be `WAT` or `GAS`. In compositional mode, the nature of the injected fluid (that is, the mole fraction of each component) must be known. This may be defined in one of the following ways:

   - the injection stream has a user specified composition, defined with the `WELLSTRE`, `WINJMIX` or `WINJORD` keyword
   - the injection stream is taken from the production of a named well or group.

   The fluid string must be one of the following:

   - **WAT**  Water.
   - **GAS**  Gas may be selected in either a black oil or a compositional run.
     - In a compositional run the composition of the injected gas is set to that of the field separator gas. However, it is generally better to specify the injected gas composition more precisely.
     - From 2007.1, in a compositional run, `GAS` is internally converted to `GV` (for the field separator) which has better reinjection and gas accounting behavior. item 113 of `OPTIONS3` can be used to recover the pre-2007.1 behavior.
   - **ST**  Implies that the molar composition of the injected fluid has been defined using the `WELLSTRE` keyword.
   - **MIX**  Implies that the molar composition of the injected fluid has been defined as a mixture using either the `WINJMIX` or `WINJORD` keywords.
   - **WV**  Implies that the injected fluid is to be taken from the vapor production of a nominated well.
GV Implies that the composition of the injected fluid is to be taken from the vapor production of a nominated group.

The injection fluid string may only be defaulted if the well is injecting gas under group control, in which case the injection stream is that defined for the group.

3. A character string is required if the composition of the injection fluid was specified using the ST, MIX, WV or GV methods.

In these cases this item is a character string specifying the name of the wellstream, mixture, well or group that defines or supplies the injection stream.

4. A character string specifying the method used to control the rate at which fluid is injected.

This must be one of the following:

WAT Inject a target water volume rate set in item 6
GAS Inject a target gas volume rate set in item 7
BHP Maintain a target BHP set in item 8
THP Maintain a target THP set in item 9
RV Inject the fluid at a target reservoir volume rate set in item 10
WG Inject the fluid at a target wet gas volume rate set in item 11
RE Reinjection or voidage replacement control, at a fraction set in item 12
TM Inject the fluid at a target total molar rate set in item 20
CW Continuous water/gas injection
GR Specifies that the well is under group control.

The character string determines the constraint to be used as an initial target for injection; other constraints entered are interpreted as limits.

*Note:* To achieve multi-component injection, use STREAM for the injection fluid and CW for the control method. This requires WELLSTRE to specify the mole fractions of the hydrocarbon gas components. Water vapor and liquid enthalpies are calculated with the quality and temperature as specified in items 21 and 22. Hydrocarbon gas enthalpies are calculated with the temperature and pressure in items 22 and 23 (pressure defaults to the water vapor pressure if unspecified). Water and hydrocarbon gas volume rates are specified with the following items 6 and 7.

5. The target/limit oil volume rate.

UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

6. The target/limit water volume rate.

UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

7. The target/limit gas volume rate.
8. The target/limit bottom hole pressure.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

9. The target/limit tubing head pressure.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

10. The target/limit reservoir volume rate.
    UNITS: rm$^3$/day (METRIC), rb/day (FIELD), rcc/hr (LAB), rm$^3$/day (PVT-M)

11. The target/limit wet gas volume rate.
    The wet gas volume injected is related to the moles injected using a unity Z-factor at standard conditions.
    UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

12. The target/limit reinjection fraction or voidage replacement fraction.
    If item 18 is set to S, item 12 is interpreted as a reinjection fraction. When this is a target, the well will attempt to inject the fluid specified in item 2 at a surface rate equal to the reinjection fraction multiplied by the water or available produced gas rate of the group or well named in item 14. The available gas rate of a group for reinjection is by default equal to the group’s gas production rate plus the rate of any advanced gas associated with the group (keyword GADVANCE) minus its fuel rate (keyword GRUPFUEL) minus its sales target (keyword GRUPSALE); however, the ordering of this definition can be altered using keyword WTAKEGAS. In the case of a reinjection fluid defined by a stream or mixture, the specified fraction of the available surface gas production rate is applied as a wet gas reinjection rate target.
    If item 18 is set to R, item 12 is interpreted as a voidage replacement fraction. When this is a target, the well attempts to inject the fluid specified in item 2 at a reservoir volume rate equal to the voidage replacement fraction multiplied by the total reservoir volume production rate of the group or well named in item 14.
    A make-up gas stream may be specified in item 17, which is mixed with the injection stream if the target gas injection rate exceeds the available gas rate.
    DEFAULT: 1.0 if item 4 is RE 0.0 otherwise

13. The type of production used to define the rate under reinjection or voidage replacement control.
    This is required for reinjection or voidage replacement control. It may be one of the following:
    W Implies the reinjection or voidage replacement depends on the production of the well named in item 14.
    G Implies the reinjection or voidage replacement depends on the production of the group named in item 14.
    DEFAULT: W if item 2 is WV
    DEFAULT: G if item 2 is GV

14. The name of the production well or group that controls the reinjection or voidage replacement rate.
This is required for reinjection or voidage replacement control. If item 13 is $W$ it should be the name of a production well. If item 13 is $G$ it should be the name of a group.

If this name is not specified, it defaults to the fluid composition source name (item 3) for injection fluids $WV$ or $GV$ in item 2.

15. Gas rate to be subtracted from the reinjection rate target.

This is only relevant when reinjecting a fraction of a named well’s production. The well in item 1 then attempts to inject at a rate equal to the reinjection fraction multiplied by the production rate of the well in item 14, minus the rate in item 15.

UNITs: $sm^3/day \ (METRIC)$, $Mscf/day \ (FIELD)$, $scc/hr \ (LAB)$, $sm^3/day \ (PVT-M)$

16. The VFP table number for the well.

The injection well VFP table must be supplied with the keyword $VFPINJ$.

This is only required if a THP target or limit is set.

17. The mnemonic for a wellstream to be used as make-up gas

This is only required if make-up gas is to be used when the injected gas rate exceeds the advance and/or available produced gas from the source group or well.

18. The way in which reinjection or voidage replacement fraction in item 12 is to be interpreted.

S Reinject a fraction of surface volume production (that is, reinjection)

R Replace a fraction of reservoir volume production (that is, voidage replacement)

DEFAULT: S

19. The stage of the separator that defines the fluid composition for injection

The vapor from any stage of the separator may act as a source of injection fluid. The default value of 0 results in the use of the vapor from the whole separator as the injection fluid.

Note that the choice of separator stage also affects the available produced gas supply rate from the source group or well (item 3) if the source is $GV$ or $WV$ (item 2), and also that of the control group or well (item 14) for reinjection.

DEFAULT: 0

20. The target or limit total molar rate.

UNITs: $kg-M/day \ (METRIC)$, $lb-M/day \ (FIELD)$, $gm-M/hr \ (LAB)$, $kg-M/day \ (PVT-M)$

21. The steam quality of the injected fluid.

(available only with the Thermal option).

If specified, the steam quality must be between 0 and 1. If specified, either the temperature or the pressure must also be specified, but not the specific enthalpy rate.

22. The temperature of the injected fluid.

(available only with the Thermal option or the Temperature option).

If specified, either the steam quality or the pressure must also be specified, but not the specific enthalpy rate.
UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

Note: To maintain a constant injection well temperature with a specified pressure, the pressure should approximately match the well’s current BHP. If the pressure is specified to be lower than the current BHP, for example, then the injection energy and hence the actual well temperature is lower than that required. If the completion block temperature is to be maintained at a constant value, then the specified pressure should be approximately that of the region around the well. In this case, the reported well temperature may be lower than the completion block temperature because the reported well temperature is calculated at the well’s BHP.

23. The pressure of the injected fluid.
   (available only with the Thermal option or the Temperature option).
   If specified, then one of (steam quality, temperature, specific enthalpy rate) must also be specified.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

24. The specific enthalpy rate of the injected fluid.
   (available only with the Thermal option).
   If specified, the pressure must also be specified and neither the steam quality nor temperature may be specified.
   UNITS: kJ/kg-M (METRIC), Btu/lb-M (FIELD), J/gm-M (LAB), kJ/kg-M (PVT-M)

Notes

See notes number 2, number 3 and number 4 in the GRUPINJE keyword documentation.

Examples

Example 1
Water is to be injected by the well at a bottom hole pressure of 4020 psia.

```
WELLINJE
WELLX Water ' ' Bhp 3* 4020 /
```

Example 2
The hydrocarbon defined by wellstream MixC is to be injected at a wet gas rate of 100 Mscf per day:

```
WELLINJE
WZ-2 Stream MixC WG 6* 100 /
```

Example 3
Well I injects all vapor from group G, minus sales gas of 1500 Mscf/day.

```
WELLINJE
I GVapour G Reinj 7* 1.0 GVapour G 1500 /
```
**Example 4**

Wells I₁ and I₂ both inject under the control of group GR₁. This group supplies the rate targets and the nature of the fluid to be injected:

```
WELSPECS
 I1 GR1 ... /
 /
WELLINJE
 I1 2* GR /
 I2 2* GR /
 /
```

**Example 5**

Well I is to inject the mixture SOLVENT, previously defined using a WELLSTRE keyword, at a rate of 150000 m³/day:

```
WELLINJE
 I Stream SOLVENT Gas 2* 150000 /
 /
```
**WELLKBHP**  Specify K-index of cell used to define BHP reference depth

This keyword is used to specify the K index in the grid of the completion cell used to define the BHP reference depth. It will overwrite the depth entered in keyword WELSPECS with the center depth of the grid block containing the first defined completion of the well in a specified layer. The well must first have a completion defined in the layer.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/). The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. The K index (layer number, counting from top) used to define the bottom hole pressure reference depth.

**Example**

```
WELLKBHP
W24 4 /
```

**Keywords**

2520
WELLLIM  Specifies well economic limits

This is an alias for WECON.
WELLOPEN  Opens a well in the SCHEDULE section

This keyword reopens an existing well, replacing it in the simulation. The normal use of this keyword is to reopen wells previously closed either automatically or using WELLSHUT.

The keyword is retained for back-compatibility; you are encouraged to use keyword WELLOPEN instead, which is compatible with both simulators.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

Example

```
WELLOPEN
  WellA1 /
  PR14   /
/
```

Keywords

```
<table>
<thead>
<tr>
<th>ECLIPSE 100</th>
</tr>
</thead>
<tbody>
<tr>
<td>x ECLIPSE 300</td>
</tr>
<tr>
<td>SPECIAL</td>
</tr>
<tr>
<td>RUNSPEC</td>
</tr>
<tr>
<td>GRID</td>
</tr>
<tr>
<td>EDIT</td>
</tr>
<tr>
<td>PROPS</td>
</tr>
<tr>
<td>REGIONS</td>
</tr>
<tr>
<td>SOLUTION</td>
</tr>
<tr>
<td>SUMMARY</td>
</tr>
<tr>
<td>x SCHEDULE</td>
</tr>
</tbody>
</table>
```
WELLOPTS  Select well solution options

WELLOPTS is an alias for NUPCOL.
WELLPROD  Specifies well production targets

This keyword specifies the production targets for a well.

The keyword is retained for back-compatibility; you are encouraged to use keyword WCONPROD instead, which is compatible with both simulators.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Well name, well name template, well list or well list template

A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. A character string defining the method of controlling the well.

This must be one of the following:

- **OIL**: Oil volume rate
- **WAT**: Water volume rate
- **GAS**: Gas volume rate
- **LIQ**: Liquid volume rate
- **COM**: Linearly combined volume rate (coefficients must be set with LINCOM)
- **BHP**: Bottom hole pressure control
- **THP**: Tubing head pressure control
- **RV**: Reservoir voidage rate
- **WG**: Wet gas rate
- **TM**: Total molar rate
- **ST**: Steam rate (available only with the Thermal option)
- **SATP**: Water saturation pressure control (available only with the Thermal option)
- **SATT**: Water saturation temperature control (available only with the Thermal option)
- **GROUP**: Specifies that the well is under group control

Only the first two characters are significant.
3. The target oil volume rate.
   UNITS: m³/day (METRIC), stb/day (FIELD), scf/hr (LAB), m³/day (PVT-M)

4. The target water volume rate.
   UNITS: m³/day (METRIC), stb/day (FIELD), scf/hr (LAB), m³/day (PVT-M)

5. The target gas volume rate.
   UNITS: m³/day (METRIC), Mscf/day (FIELD), scf/hr (LAB), m³/day (PVT-M)

6. The target liquid (oil+water) volume rate.
   UNITS: m³/day (METRIC), stb/day (FIELD), scf/hr (LAB), m³/day (PVT-M)

7. The target or limiting bottom hole pressure.
   UNITS: bara (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)

8. The target tubing head pressure.
   UNITS: bara (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)

9. The target or limit reservoir volume rate
   Volume rate is the reservoir volume of the fluids produced or injected when at the well reference pressure. The reference pressure is usually the average hydrocarbon pressure in the field. However, this may be set to the average pressure in a nominated fluid-in-place region using item 13 in WELSPECS.
   UNITS: m³³/day (METRIC), rb/day (FIELD), rcc/hr (LAB), m³³/day (PVT-M)

10. The target wet gas volume rate.
    The volume of wet gas is required. Given a number of moles, this can be obtained as \( V = \frac{nRT}{P} \), where \( T \) and \( P \) are standard pressure and temperature, \( R \) is the gas constant, \( V \) is the volume rate and \( n \) is the molar rate. See "Units" in ECLIPSE Technical Description for further details.
    If group control is specified, the well target need not be specified.
    UNITS: m³³/day (METRIC), Mscf/day (FIELD), scf/hr (LAB), m³³/day (PVT-M)

11. The VFP table index for this well.
    Only required if a THP target or limit is to be specified.

12. Artificial lift quantity, for use in THP calculations.
    See keyword VFPPROD.
    UNITS: These depend on what the ALQ was meant to represent when the table was created. Refer to section "Record 6:ALQ values" in the description of keyword VFPPROD.
    DEFAULT: 0.0

13. The target or limit total molar rate.
    UNITS: Kg-M/day (METRIC), lb-M/day (FIELD), gm-M/hr (LAB), Kg-M/day (PVT-M)

14. The target or limit steam rate, Cold Water Equivalent, (available only with the Thermal option).
    UNITS: m³³/day (METRIC), stb/day (FIELD), scf/hr (LAB), m³³/day (PVT-M)
15. The pressure offset $\Delta P$ for saturation pressure control, (available only with the Thermal option). The bottom hole pressure is constrained by $BHP \geq Psat + \Delta P$ where $Psat$ is the maximum saturated water pressure in all the producing completion cells.

UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

16. The temperature offset $\Delta T$ for saturation pressure control, (available only with the Thermal option). The bottom hole pressure is constrained by $BHP \geq Psat (T + \Delta T)$ where $T$ is the maximum temperature in all the producing completion cells.

UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

17. The target/limit linearly combined rate

UNITS: determined by keyword LCUNIT

Examples

Example 1

The well is set to a target oil volume rate of 1000 stb/day, with a bottom hole pressure limit of 4511 psia.

```
WELLPROD
P Oil 1000.0 3* 4511.0 /
```

Example 2

The well is placed under group control, but with a minimum bottom hole pressure limit of 400 psia.

```
WELLPROD
P GR 4* 400 /
```
WELLSHUT  Shuts a well in the SCHEDULE section

This keyword shuts an existing well, effectively removing it from the simulation.

The keyword is retained for back-compatibility; you are encouraged to use keyword WELOPEN instead, which is compatible with both simulators.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

Example

```plaintext
WELLSHUT
  WellA1 /
  PR14 /
/
```
WELLSMA  Calculate connection critical drawdowns for sanding and take action

This keyword requests a Sand Management Advisor (SMA) calculation of all connection critical drawdowns, and that appropriate action be taken if the completion drawdown exceeds a percentage of this critical drawdown. This keyword with its calculation is not available commercially.

A license for Schlumberger staff to run WELLSMA can be obtained by contacting ECLIPSE support.
**WELLSPEC**  **Introduces a new well**

This keyword introduces a new well, specifying the name and position of the well head, the BHP reference depth and the separator used.

The keyword is retained for back-compatibility; you are encouraged to use keyword `WELSPECS` instead, which is compatible with both simulators.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. The name of the well.
   A character string of up to 8 letters. This must be specified and non-blank.

2. The name of the group to which the well belongs.
   **DEFAULT:** `FIELD`

3. The I-index of the header.

4. The J-index of the header

5. The bottom hole pressure reference depth.
   **UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   If this quantity is defaulted, the reference depth is set equal to the center depth of the grid block containing the topmost connection in the well

6. The name of the separator to which the well is connected.
   A separator need only be nominated in the compositional case for production wells. The separator nominated must have been previously defined using the `SEPCOND` keyword.

   **Note:** If a separator is not specified here or with `WSEPCOND`, and the well can trace its parentage to a group which has a default separator specified (see `SEPCOND`), then this will be used. If this is not the case, the separator conditions specified with `FIELDSEP` are used. If `FIELDSEP` is not entered, a single flash to standard conditions is used.

7. Fluids-in-place number to obtain reservoir conditions used in calculating a well’s voidage rate. This is only required if the crossflow well model is being used.
   - If 0 is entered, the average hydrocarbon conditions in the field are used.
   - If a negative value is entered, the region number is set equal to the fluids-in-place region number of the lowest grid block in which the well is completed.
   - If a positive value is entered, this is interpreted as the required fluids-in-place region number. A warning message is printed if the well is not completed in a grid block within that region.
DEFAULT: 0

Example

Well P, in group G, is located in cell I=2, J=2, with a BHP reference depth of 8345 ft, and feeds separator SepA1.

```
WELLSPEC
  P G 2 2 8345 SepA1 /
```
WELLSTRE  Set composition of injection stream

When, in a compositional or thermal run, oil or gas is injected via a stream, this keyword may be used to specify the composition of the injected stream.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values of 0.0. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. The name of the well stream.
   The maximum number of wellstreams allowed is governed by item 6 of keyword WELLDIMS.

2. $x_1$, the mole fraction of the first component. User defined arguments can be set with keywords UDQ and UDADIMS.

3. $x_2$, the mole fraction of the second component. User defined arguments can be set with keywords UDQ and UDADIMS.

...

...

...

$N_c + 1$ $x_{N_c}$, the mole fraction of the $N_c^{th}$ component. User defined arguments can be set with keywords UDQ and UDADIMS.

$N_c$ mole fractions should be specified, where $N_c$ is the number of components in the run. The values are checked using the condition that the sum of the mole fractions must be 1.0.

Once the gas wellstream is defined, it may be used in the WINJGAS or GINJGAS keyword, for example, to specify an injected composition, or used to define a make-up gas.

Once an oil wellstream is defined, it may be used in the WINJOIL keyword, for example, to specify an injected composition.

Example

```
WELLSTRE
'MixB' 0.5 0.3 0.1 0.05 0.05 /
/
```

Keywords

2531
WELLSTRW  Set composition of injection water stream

This keyword may be used to specify the composition of an injected water stream when using the multi-component water model, or when using any of the CO2STORE, GASWAT or GASSOL options.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values of 0.0. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. The name of the well stream.
   
   The maximum number of wellstreams allowed is governed by item 6 of keyword WELLDIMS.

2. $x_1$, the mole fraction of the first component.

3. $x_2$, the mole fraction of the second component.

...  

...  

$N_w + 1 x_{N_w}$, the mole fraction of the $N_w$th component.

$N_w$ mole fractions should be specified, where $N_w$ is the number of components in the run.

**Note:** If GASSOL is present, $NCMFW$ components and water are present in the aqueous phase; hence $N_w$ is the number of components in the aqueous phase $NCMFW$ followed by the water fraction. If CO2STORE or GASWAT is present $N_w$ is the number of allowed components in the CO2STORE or GASWAT run.

The values are checked using the condition that the sum of the mole fractions must be 1.0.

Once the wellstream is defined, it may be used in the WINJW keyword to specify an injected composition.

See "Water components" in the ECLIPSE Technical Description for further details of the water component option.

**Example**

```
WELLSTRW
   Fresh 1.0 0.0/
   Brine 0.0 1.0/
```
**WELLTARG**  Change well target or limit

WELLTARG is an alias for WELTARG.

<table>
<thead>
<tr>
<th>ECLIPSE 100</th>
<th>ECLIPSE 300</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>SPECIAL</td>
</tr>
<tr>
<td>RUNSPEC</td>
<td>GRID</td>
</tr>
<tr>
<td>EDIT</td>
<td>PROPS</td>
</tr>
<tr>
<td>REGIONS</td>
<td>SOLUTION</td>
</tr>
<tr>
<td>SUMMARY</td>
<td>SCHEDULE</td>
</tr>
</tbody>
</table>

Keywords

2533
Well temperature cutback

The data consists of tables of well names and the temperature cutback table number to be used for that well. Each table is terminated by a slash (/).

Each table consists of 2 columns of data.

Column:

1. The well name.
   This well name should already have been specified.

2. The corresponding cutback table number.
   A table number of zero means that no cutback is performed for this well.
   The table number supplied should be less than or equal to the number of well cutback tables in the run, specified with the TCBDIMS keyword in the RUNSPEC section.

DEFAULT: 0

This keyword can be repeated at a later timestep to assign different tables to each well.

See also the keywords WELLTCBT and WELLTCBC.

Example

```
-- well P1 uses TCB table 2
-- well P2 uses TCB table 1
WELLTCB
  P1 2 /
P2 1 /
```

Keywords

ECLIPSE Reference Manual

2534
WELLTCBC  Well temperature cutback critical temperature

This keyword defines a temperature that is used as a threshold temperature by the WELLTCB keyword. The current value of the well temperature cutback will remain unchanged until the well temperature goes beyond the limit given in WELLTCBT by more than this threshold value.

The purpose of this keyword is to try to prevent repeated changes in the well cutback value. These can arise if the well temperature is on the borderline between two cutback values.

Argument:

1. The well name.
   This well name should already have been specified.

2. The critical or threshold temperature.
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

Example

```
WELLTCBC
P1 5 /
```
WELLTCBT  Well temperature cutback table

The data consists of tables of temperature cutback factors.

Each table is terminated by a slash (/).

Each table consists of 2 columns of data.

Column:

1. The temperature.

   Values should be greater than 0 and should increase monotonically down the column.

   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

2. The corresponding cutback factor.

   Values should be between 0 and 1 and should decrease down the column. The first value in the column should be 1.

The number of tables supplied should be equal to the number of well cutback tables in the run, specified with the TCBDIMS keyword in the RUNSPEC section or defaulted to 1.

The number of rows should be less than or equal to the maximum number of rows of data specified with TCBDIMS.

The cutback factor is used to multiply the maximum or target oil rate, gas rate, water rate, liquid rate, voidage rate, and hydrocarbon rate for each well specified in the WELLTCB keyword.

The cutback factor to be used for the next timestep is not interpolated between the values specified in the tables. The cutback factor specified at a particular temperature is applied until the next highest temperature is reached. In the example below, for the first table, a factor of 1 (that is no cutback) applies for all temperatures up to 140, then a factor of 0.95 applies between 140 and 210, then a factor of 0.85 from 210 to 500, then a factor of 0.4 for any temperature above 500.

Note: These tables are not stored in the Restart files. However, a well that is linked to a table in the base run (using WELLTCB) is still linked to the table in the restart run. Therefore, the tables should be redefined in the restart run.

Example

```plaintext
-- 2TCB tables
WELLTCBT
  126 1
  140 .95
  210 .85
  500 .4
/
  135 1
  300 .6
  600 .3
/```

Keywords

2536
**WELLWAG**  
**Specifies WAG well injection targets**

This keyword specifies a WAG (water and gas) injection mode for a well. A *WCONINJE* keyword must previously have been entered to define the injection fluids, the required targets and limits for example. The initial injection mode may be set to water or gas; it is overruled by the *WELLWAG* mode.

Note that if the injection rates are re-specified with another *WCONINJE* keyword, the *WELLWAG* data must be entered again (otherwise the well returns to normal single phase injection). This need not be done if *WELTARG* is used to change the rates.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. **Well name, well name template, well list or well list template**
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the *ECLIPSE Technical Description* for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword *WLIST*.

2. **A character string specifying the type of WAG cycle.**
   
   Must be one of:
   
   T  Time  
   M  Months  
   Y  Years  
   
   DEFAULT: T

3. **The first fluid to be injected in the WAG sequence.**
   
   This may be W or G.

4. **The injection period for the fluid identified in item 3.**

5. **This item identifies the second fluid to be injected.**
   
   At present only water and gas cycling is supported, and this defaults to G if item 3 is W, and to W if item 3 is G.

6. **The injection period for the fluid identified in item 5.**

7. **Injection well VFP table number for the first fluid to be injected.**
   
   (See keyword *VFPINJ*)
   
   Set this to zero if no THP calculations are required and the limiting THP value has been defaulted.
   
   DEFAULT: Value previously set for this well.

8. **Injection well VFP table number for the second fluid to be injected.**

---

**Keywords**

2537
(See keyword VFPINJ.)
Set this to zero if no THP calculations are required and the limiting THP value has been defaulted.
DEFAULT: Value previously set for this well.

9. Control mode for first fluid to be injected.
   RATE   Control by surface flow rate target (item 5 of keyword WCONINJE or keyword WELTARG)
   BHP    Control by BHP target
   THP    Control by THP target
DEFAULT: RATE

10. Control mode for second fluid to be injected.
    RATE   Control by surface flow rate target (item 5 of keyword WCONINJE or keyword WELTARG)
    BHP    Control by BHP target
    THP    Control by THP target
DEFAULT: RATE

11. BHP target or upper limit for first fluid to be injected
    UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
    DEFAULT: Value previously set by keyword WCONINJE or keyword WELTARG

12. BHP target or upper limit for second fluid to be injected
    UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
    DEFAULT: Value previously set by keyword WCONINJE or keyword WELTARG

13. THP target or upper limit for first fluid to be injected
    UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
    DEFAULT: Value previously set by keyword WCONINJE or keyword WELTARG

14. THP target or upper limit for second fluid to be injected
    UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
    DEFAULT: Value previously set by keyword WCONINJE or keyword WELTARG

Examples

Example 1
Inject gas with composition defined by wellstream Stream at a rate of 30000 Mscf/day, and water at a rate of 45000 stb/day, with a bottom hole pressure limit of 4500 psia, alternating with a 91.25 day period, starting with water injection.

```
WELLINJE
I Stream Solvent Gas 1* 45000 30000 4500 /
```
Example 2

```
WCONINJE
I GAS OPEN RATE 30000 1* 4500 /
/
WINJGAS
I STREAM SOLVENT /
/
WELTARG
I WRAT 45000 /
/
WELLWAG
I T W 91.25 G 91.25 /
/
```
WELMOVEL  Moves a well into an LGR at a restart

WELMOVEL allows a well which was specified in the global grid to be moved into a local grid. This keyword is intended for situations where you wish to add a local grid around a preexisting well at a restart. (that is, the well existed in the global grid in the initial base run but is to be placed in a new local grid at the restart). If this keyword is used then all the previously specified connection data for the well are deleted. The connection data for these wells must then be re-specified using the keyword COMPDATL. All the well properties such as group membership, rates and cumulative totals are however preserved.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated early with a slash (/); the remaining items assume their default values.

The set of records must end with a blank record, containing only a slash (/).

1. Well name (Up to 8 characters)
2. Name of the local refined grid in which the well is located
   If the well is completed in an amalgamation of local grids (see keyword AMALGAM), the name of any local grid in the amalgamation may be used here.
3. I - location of well head, in the coordinate system of the local grid
4. J - location of well head, in the coordinate system of the local grid

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

**Note:** The cumulative totals for wells is not the same as the sum of the cumulative totals for all the connections after this keyword is used, as the well totals contain the cumulative totals for the connections that were deleted by the use of this keyword.

**Note:** This keyword was intended to provide additional functionality for the Near Wellbore Modeling tool and should only be used with caution when the data set has not been generated by this product. The keyword should be placed before the first occurrence of any COMPDAT or COMPDATL keywords in the SCHEDULE section of a restart run.

**Example**

```
WELMOVEL
  PROD1  LGR1  6 8 /
  PROD2  LGR2  5 9 /
/
```

Keywords
Shuts or reopens wells or well connections

This keyword can be used to shut or reopen wells or well connections, without having to specify the rest of the well control data or connection data. To shut or reopen a well, leave items 3 - 7 defaulted. To shut or reopen individual well connections, enter their I,J,K location in items 3 - 5, or enter their completion number in items 6 and 7. Completion numbers are set in keyword COMPLUMP.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated after item 1 or item 2. The remaining items assume their default values.

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Open/shut flag for the well or connections
   OPEN Well or connections open to flow
   SHUT Well or connections completely isolated from formation
   STOP Well stopped off above formation. (If the record applies to a connection, it has the same effect as SHUT.)
   AUTO Well or connections initially SHUT but open automatically. (See keyword WCONPROD item 2 for an explanation of AUTO applying to wells, and keyword COMPDAT item 6 for connections).
   POPN The well opens only if it is "priority-closed", that is temporarily closed under Prioritization control (see "Prioritization - an alternative to guide rate control" in the ECLIPSE Technical Description). The well remains closed if it has been closed for any other reason. This may be useful when changing the group control method from Prioritization to Guide Rate control.

   DEFAULT: OPEN

3. I - location of connecting grid block(s)
   DEFAULT: Negative

4. J - location of connecting grid block(s)
   DEFAULT: Negative

5. K - location of connecting grid block(s)
   DEFAULT: Negative

6. Number of first completion in range
   DEFAULT: Negative

7. Number of last completion in range

Keywords

2541
DEFAULT: Negative

If items 3 - 7 are all defaulted, the Open/Shut/Stop command applies to the well, leaving the connections unchanged. An attempt to open a well with no connections open results in an error condition.

If any of items 3 - 7 are set zero or positive, the command applies to the eligible connections of the well, leaving the status of the well itself unchanged. The eligible connections are those with I,J,K locations matching the specified location indices in items 3 - 5 and a completion number in the range specified by items 6 - 7. A zero or negative value for a location index (I, J or K) matches any value of that location index. Similarly a zero or negative value for a completion range lower/upper end point indicates that there is no lower/upper bound on the completion numbers.

To open a well and one or more of its connections requires two records, one to operate on the well and the other to operate on the connections.

End the data record with a slash (/).

End the set of records with a blank record containing just a slash.

Note: If this keyword is used to shut a well during a history matching simulation, the actual production or injection rates are set to zero from this point in time, but all historical well rates remain unaltered. If the well was actually shut at this point in time, then the WCONHIST or WCONINJH keywords should be used to shut the well and thus zero the historical well rates.

Example

<table>
<thead>
<tr>
<th>Record</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>WELOPEN</td>
<td>opens all shut or stopped wells in the field</td>
</tr>
<tr>
<td>PROD3 SHUT</td>
<td>shuts well PROD3, if it is open</td>
</tr>
<tr>
<td>'*'</td>
<td>opens all well connections in layer 3</td>
</tr>
<tr>
<td>PROD4 OPEN</td>
<td>opens all connections of well PROD4</td>
</tr>
<tr>
<td>PROD5 SHUT</td>
<td>shuts connection(s) to layer 4 in well PROD5</td>
</tr>
<tr>
<td>PROD6 SHUT</td>
<td>shuts connection in completions 4, 5 and 6 in PROD6</td>
</tr>
</tbody>
</table>

End
WELOPENL  Shuts or reopens wells or well connections within local grids

This keyword can be used to shut or reopen wells or well connections in local grids, without having to specify the rest of the well control data or connection data. The connections must first have been defined with the keyword COMPDATL. WELOPENL performs the same role as WELOPEN does, but the name of the local grid containing the connections specified in the record should be entered in item 2 of WELOPENL.

To shut or reopen a well, leave items 4-8 defaulted. Well connections can be shut or reopened either according to their I,J,K location (items 4 - 6) or according to their completion number (items 7 and 8). Completion numbers are set in keyword COMPLMPL.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated after item 1 or item 2. The remaining items will assume their default values.

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template.
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Name of local grid containing the connections to be opened or closed
   If this item is defaulted or set to a blank string, the local grid name will be set to the well’s local grid named in keyword WELSPECL. (In ECLIPSE 100, the local grid name should not be defaulted if the local grid is amalgamated, unless items 4-8 have been defaulted.)

   The local grid name may also be defaulted if the connection locations in items 4, 5 and 6 are all defaulted, as the action in item 3 then applies to the well itself (or to specific completions within it if items 7 and 8 are set, and ECLIPSE 100 already knows which local grid they belong to).

   DEFAULT:’ ’

3. Open/shut flag for the well or connections
   OPEN  Well or connections open to flow
   SHUT  Well or connections completely isolated from formation
   STOP  Well stopped off above formation, allowing crossflow through the well bore if two or more connections are open (If the record applies to a connection, it has the same effect as SHUT).

   ECLIPSE 100 only
   AUTO  Well or connections initially SHUT but open automatically. (See keyword WCONPROD item 2 for an explanation of AUTO applying to wells, and keyword COMPDAT item 6 for connections).

   POPN  The well opens only if it is “priority-closed”, that is temporarily closed under Prioritization control (see "Prioritization - an alternative to guide rate control" in the ECLIPSE Technical Description).
**Description**). The well remains closed if it has been closed for any other reason. This may be useful when changing the group control method from Prioritization to Guide Rate control.

**DEFAULT:** OPEN

4. I - location of connecting grid block(s)
   **DEFAULT:** Negative

5. J - location of connecting grid block(s)
   **DEFAULT:** Negative

6. K - location of connecting grid block(s)
   **DEFAULT:** Negative

7. Number of first completion in range
   **DEFAULT:** Negative

8. Number of last completion in range
   **DEFAULT:** Negative

If items 4 - 8 are all defaulted, the Open/Shut/Stop command applies to the well, leaving the connections unchanged. An attempt to open a well with no connections open will result in an error condition.

If any of items 4 - 8 are set zero or positive, the command applies to the eligible connections of the well, leaving the status of the well itself unchanged. The eligible connections are those with I,J,K locations matching the specified location indices in items 4 - 6 and a completion number in the range specified by items 7 - 8. A zero or negative value for a location index (I, J or K) matches any value of that location index. Similarly a zero or negative value for a completion range lower/upper end-point indicates that there is no lower/upper bound on the completion numbers.

To open a well **and** one or more of its connections requires two records, one to operate on the well and the other to operate on the connections.

End the data record with a slash (/).

End the set of records with a blank record containing just a slash.

**Example**

```
WELOPENL
-- Open all shut or stopped wells whose names begin with P in the field 'P*' /

-- Shut well PROD3, if it is open
PROD3 1* SHUT /

-- Shut all wells having connections in local grid CARF1
'***' CARF1 SHUT /

-- (and any other local grids amalgamated with it)
'***' CARF2 OPEN 0 0 3 /
```

**Keywords**
-- Open all connections in local grid CARF3 for well PROD4
PROD4  CARF3  OPEN  0  0  0 /
-- Shut all connections in local grid CARF4 for well PROD4
PROD4  CARF4  SHUT  0  0  0 /
-- Shut connections in layer 4 of local grid CARF2 for well PROD5
PROD5  CARF2  SHUT  0  0  4 /
-- Shut connection in completions 4, 5 and 6 in well PROD6
PROD6  1*  SHUT  0  0  4  6/
WELPI

Sets well productivity/injectivity index values

This keyword can be used to set the productivity/injectivity index (PI) value of one or more wells to a specified value. The well connection properties must be defined in the usual manner, with keyword COMPDAT, but the connection transmissibility factors are adjusted proportionately so that the calculated PI of the well equals the specified value. The PI is calculated using the phase and drainage radius defined in items 6 and 7 of keyword WELSPECS. The definition of the PI and its relationship to the well’s connection transmissibility factors is discussed in “The productivity index” in the ECLIPSE Technical Description. Note that the calculation is not suitable for application to horizontal wells.

The adjustment of the connection factors is performed at the time the WELPI keyword is entered, using the current grid block solution. If the fluid mobilities in the grid blocks subsequently change, the well’s actual PI also changes.

The P.I. value should not be set for gas wells which have a D-factor or use a special inflow equation (item 8 in WELSPECS). In these circumstances the gas phase P.I. vary with the well’s BHP.

Under some circumstances the P. set for a newly opened gas injector are not exact due to the composition of the injected gas not being accurately known during the calculation.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Steady-state productivity/injectivity index value of the well(s)

   UNITS: sm$^3$/day/bars (METRIC), stb/day/psi or Mscf/day/psi (FIELD), scc/hr/atm (LAB), sm$^3$/day/atm (PVT-M)

   This item must be supplied and cannot be defaulted.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Note: When the WELPI keyword is read, the connections that are currently defined for the well will have their connection factors scaled to give the well the specified PI at the current grid block conditions. If at a later time new connections are introduced, the connection factors are not scaled again unless WELPI is reentered, and so the PI of the well changes. Moreover, if an existing connection is later re-specified with COMPDAT, its connection factor is reset from the COMPDAT data, overwriting its earlier scaled value. Consequently, care should be taken when changing the connection data after entering WELPI.

CAUTION: Keywords WELPI and WPIMULT should not both be used on the same well at the same time, unless there is an intervening TIME, TSTEP or DATES keyword.
Example

```
WELPI
  PROD1  10.2 /
  PROD2  14.3 /
/
```
WELPRI

Sets well priority numbers

This keyword can be used to set the priority numbers for one or more production wells. The priority numbers set here will override the values calculated using the priority equation (see keyword PRIORITY). If the keyword is not used, the priority numbers for all wells will be calculated from the priority equation.

The keyword PRIORITY must be used before using this keyword.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. First priority number for the well(s)
   
   If the priority number is greater than zero, this number overrides the value calculated from the first priority equation.
   
   If the number is negative, the well priority number is calculated from the first priority equation.
   
   The number must not be zero.
   
   DEFAULT: Negative
   
   Priority numbers do not have any units.

ECLIPSE 300

Note: Currently, ECLIPSE 300 has a single priority equation, and well priorities cannot be scaled.

ECLIPSE 100 only

3. First priority scaling factor.
   
   The well’s first priority number (either set in item 2 or calculated from the first priority equation) is multiplied by the number specified here.
   
   DEFAULT: 1.0

ECLIPSE 100 only

4. Second priority number for the well(s)
   
   If the priority number is greater than zero, this number overrides the value calculated from the second priority equation.
   
   If the number is negative, the well’s second priority number is calculated from the second priority equation.
   
   The number must not be zero.
   
   This item must be defaulted if a second priority equation has not been defined.
   
   DEFAULT: Negative
   
   Priority numbers do not have any units.

ECLIPSE 100 only

5. Second priority scaling factor.
The well’s second priority number (either set in item 4 or calculated from the second priority equation) is multiplied by the number specified here.

DEFAULT: 1.0

This item must be defaulted if a second priority equation has not been defined.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Example

```
WELPRI
 'PROD*' -1.0  /  
 PROD4  1*   2.0  1*   2.0 /  (ECLIPSE 100 only)
 PROD5  100  1*   100  /  (ECLIPSE 100 only)
 /  
```
WELSEGS **Defines the segment structure of a multisegment well**

This keyword is used to designate a well to be a multisegment well, and to define the segment structure of the well. The well must previously have been introduced with the WELSPECS keyword (or WELSPECL if the well is situated in a local grid refinement).

The keyword can only be used on one well at a time, so it must be specified more than once if there is more than one multisegment well.

The WELSEGS keyword may be re-entered for a particular well later in the run, if you wish to change its segment structure (for example to add a new branch). Only the new or modified segments need to be entered; the properties of all the other segments are retained. But if the segment structure is changed, you should then reenter the COMPSEGS (or COMPSEGL) data to reallocate the grid block connections to the new segments.

Multisegment wells are described in detail in "Multisegment wells" in the ECLIPSE Technical Description.

The keyword is followed by a number of data records to describe the segment properties and geometry of the (possibly multibranched) well. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

The data items in each record are:

**Record 1**

The first record identifies the well, sets some general flags, and sets the properties of its top segment, the one nearest the wellhead. This segment is labeled segment number 1.

1. Name of the well
2. Depth of the nodal point of the top segment
   
   ECLIPSE takes this value to be the BHP reference depth of the well. BHP reference depth data entered elsewhere are ignored for multisegment wells.

   **UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

   This item must be entered the first time each multisegment well is declared in WELSEGS. On subsequent entries of WELSEGS it may be defaulted to keep its original value. The depth should not normally be changed during the simulation.

3. Length down the tubing to the nodal point of the top segment

   This item fixes the position of the top segment’s node (the BHP reference point) relative to the ‘zero tubing length reference point’, from which all the tubing length values are measured. There are no pressure losses calculated across this segment, as the multisegment well model does not calculate pressure losses above the nodal point of the top segment. (Pressure losses between the BHP reference depth and the tubing head should normally be handled by VFP tables in the usual way.)

   A zero value entered here implies that tubing lengths are measured from the BHP reference point in the top segment.

   **UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

   **DEFAULT:** 0.0 (the first time a multisegment well is declared), no change (on any subsequent entries).
4. Effective wellbore volume of the top segment
   This quantity is used to calculate wellbore storage effects in the top segment.
   UNITS: m$^3$ (METRIC), ft$^3$ (FIELD), cc (LAB), m$^3$ (PVT-M)
   DEFAULT: 1.0E-5, giving negligible storage effects (the first time a multisegment well is declared),
   no change (on any subsequent entries).

5. Type of tubing length and depth information entered in the subsequent records of this keyword

   INC  The length and depth values entered in items 5 and 6 of the subsequent records refer to the
   incremental changes of these quantities along each segment.

   ABS  The length and depth values entered in items 5 and 6 of the subsequent records refer to the
   absolute values of the tubing length and depth at the segment nodes.

   DEFAULT: There is no default; this item must be set.

6. Components of the pressure drop to be included in the calculation for each of the well’s segments:

   HFA  Hydrostatic + friction + acceleration

   HF-  Hydrostatic + friction

   H-   Hydrostatic only. This item must be contained within quotes.

   For segments whose friction+hydrostatic pressure drops are obtained from a VFP table (item 5 in
   keyword WSEGTABL set to FH), the total pressure drop is treated as friction; the hydrostatic
   component is not separated out. If HFA is selected here, the acceleration pressure drop is calculated
   separately using a homogeneous flow model and added to the pressure drop interpolated from the VFP
   table. In this case, ideally the VFP table should not contain the acceleration pressure loss, as it is then
   accounted twice; but this component of the pressure loss is generally very small. (See "Pressure drop
   components" in the ECLIPSE Technical Description.) If this well contains flow control devices
   specified with keywords WSEGAICD, WSEGSICD, WSEGLABY or WSEGVALV, or looped flow paths
   specified with keyword WSEGLINK, then this item must be set so that friction is included.

   DEFAULT: HFA

7. Default Multiphase flow model for the well’s segments

   HO  Homogeneous flow; the phases all flow with the same velocity.

   DF  A Drift Flux Model, which allows for slip between the phases. The slip is a combination of
   profile slip (a velocity ratio) and local slip (a relative vertical velocity). At low flow rates the
   local slip can cause countercurrent flow, where the heavy and light phases flow in opposite
   directions. (See "The drift flux slip model" in the ECLIPSE Technical Description.)

   The first time each segment is defined with WELSEGS, its multiphase flow model is set according to
   the above choice. The keyword WSEGFMOD may subsequently be used to change the flow model for
   specific segments or reset the model’s parameters for specific segments.

   If WELSEGS is entered again for the same well, this item does not affect the flow model for segments
   that have been defined in an earlier entry. The exception to this is when HO is entered; any segments
   that use the drift flux model change to homogeneous flow.
ECLIPSE 100

**Note:** DF must be selected here if any of the well's segments are to use the Drift Flux Model. The top segment (segment 1) always uses the Homogeneous Flow Model; there are no pressure losses calculated along this segment.

Individual segments may also be designated to have their pressure drops determined from VFP tables instead of a built-in Flow Model (see keyword WSEGTABL).

**DEFAULT:** HO

8. X coordinate of the nodal point of the top segment, relative to the origin of the grid

The X and Y coordinates of the segment nodes are stored on the restart or graphics files to enable graphics applications to produce plots of the well trajectory. The data is not used for any other purpose and may be left defaulted if trajectory plots are not required.

**UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

**DEFAULT:** 0.0 (the first time a multisegment well is declared), no change (on any subsequent entries).

9. Y coordinate of the nodal point of the top segment, relative to the origin of the grid

**UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

**DEFAULT:** 0.0 (the first time a multisegment well is declared), no change (on any subsequent entries).

10. **Thermal Option only** Cross-sectional area of the pipe wall used in thermal conductivity calculation

(see note 3).

**UNITS:** m² (METRIC), ft² (FIELD), cm² (LAB), m² (PVT-M)

**DEFAULT:** 0.0

11. **Thermal Option only** Volumetric heat capacity of the pipe wall.

**UNITS:** kJ/m³/K (METRIC), Btu/ft³/°R (FIELD), J/cc/K (LAB), kJ/m³/K (PVT-M)

**DEFAULT:** 0.0

12. **Thermal Option only** Thermal conductivity of the pipe wall.

**UNITS:** kJ/m/day/K (METRIC), Btu/ft/day/°R (FIELD), J/cm/hr/K (LAB), kJ/m/day/K (PVT-M)

**DEFAULT:** 0.0

End the data record with a slash (/).

---

### Each subsequent record

The subsequent records define the properties of all the well's segments except for the top segment. The segments must be numbered between 2 and NSEGXM (entered in the RUNSPEC section keyword WSEGDIMS). Segments may be entered in any order, and neighboring segments need not be numbered consecutively; ECLIPSE reorders them internally. A contiguous set of segments having the same properties can be entered together in a single record as a range. The segments within a range must have consecutively increasing numbers.

1. Segment number at the start of the range (nearest the top segment)

---

**Keywords**

2552
An integer between 2 and NSEGMX.

2. Segment number at the far end of the range
An integer between 2 and NSEGMX, not less than item 1.

The remaining items in this record set the properties of the segments in this range. To enter properties for a single segment, set items 1 and 2 equal.

3. The branch number
An integer between 1 and NLBRMX (set in the RUNSPEC section in keyword WSEGDIMS).

Segments situated on the main stem must have a branch number of 1. Lateral branches should be numbered between 2 and NLBRMX. Lateral branches may also have subordinate branches; in this case each branch must have a higher branch number than its parent branch.

4. Number of the segment to which the segment at the start of the range (item 1) is joined, that is its neighbor in the direction of the wellhead. This is its outlet segment; the convention being that production flows are positive.

A node that is a branch point in a multilateral well can be the outlet segment of two or more segments on different branches.

5. If item 5 of Record 1 is INC:

The length of each segment in the range, that is the distance between its nodal point and that of its neighbor in the direction of the wellhead.

Or, if item 5 of Record 1 is ABS:

The length down the tubing (from the zero tubing length reference point) at the nodal point of the last segment in the range. ECLIPSE divides the length between this point and the node of the range’s outlet segment (item 4) into n equal lengths, where n is the number of segments in this range.

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT–M)

6. If item 5 of Record 1 is INC:

Depth change along each segment in the range, that is the depth of its nodal point minus the depth of its outlet segment’s nodal point.

Or, if item 5 of Record 1 is ABS:

The depth of the nodal point of the last segment in the range. ECLIPSE calculates the depths of any other segments in this range by interpolating between this depth and the depth of the range’s outlet segment (item 4), linearly according to their length down the tubing.

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT–M)

7. Tubing internal diameter (or the equivalent diameter for annular cross-sections)

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT–M)

8. The effective absolute roughness of the tubing.

This quantity is used to calculate the Fanning friction factor.

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT–M)

9. Cross-sectional area for fluid flow
10. Volume of the segment

**UNITS:** m³ (METRIC), ft³ (FIELD), cm³ (LAB), m³ (PVT-M)

**DEFAULT:** \( \pi \left( \frac{D}{2} \right)^2 / 4 \)

11. If item 5 of Record 1 is **INC**:

The length of each segment in the range projected onto the X-axis, that is the X coordinate of its nodal point minus that of its neighbor in the direction of the wellhead.

Or, if item 5 of Record 1 is **ABS**:

X coordinate of the nodal point of the last segment in the range, relative to the origin of the grid. ECLIPSE calculates the X coordinates of any other segments in this range by interpolating between this value and the X coordinate of the range’s outlet segment (item 4), linearly according to their length down the tubing.

The X and Y coordinates of the segment nodes are stored on the restart or graphics files to enable graphics applications to produce plots of the well trajectory. The data is not used for any other purpose and may be left defaulted if trajectory plots are not required.

**UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

**DEFAULT:** 0.0

12. If item 5 of Record 1 is **INC**:

The length of each segment in the range projected onto the Y-axis, that is the Y coordinate of its nodal point minus that of its neighbor in the direction of the wellhead.

Or, if item 5 of Record 1 is **ABS**:

Y coordinate of the nodal point of the last segment in the range, relative to the origin of the grid. ECLIPSE calculates the Y coordinates of any other segments in this range by interpolating between this value and the Y coordinate of the range’s outlet segment (item 4), linearly according to their length down the tubing.

**UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

**DEFAULT:** 0.0

13. Cross-sectional area of the pipe wall used in thermal conductivity calculation (see note 3). For tubing with circular cross-sections the area can be calculated using the formula

\[ A = \pi \cdot \left( r_o^2 - r_i^2 \right) \]

where \( r_o \) is the outer radius of and \( r_i \) is the inner radius. (ECLIPSE 300 thermal option only.)

**UNITS:** m² (METRIC), ft² (FIELD), cm² (LAB), m² (PVT-M)

**DEFAULT:** item 10 of Record 1

14. Volumetric heat capacity of the pipe wall (ECLIPSE 300 thermal option only).

**UNITS:** kJ/m³/K (METRIC), Btu/ft³°R (FIELD), J/cc/K (LAB), kJ/m³/K (PVT-M)

**DEFAULT:** item 11 of Record 1

15. Thermal conductivity of the pipe wall (ECLIPSE 300 thermal option only).
UNITs: kJ/m/day/K (METRIC), Btu/ft/day/°R (FIELD), J/cm/hr/K (LAB), kJ/m/day/K (PVT-M)

DEFAULT: item 12 of Record 1

End the data record with a slash (/).

End the set of data records with a slash.

Notes

1. Multisegment well data can be complicated to enter. It is advisable to check your data by examining
the output in the Print file.

ECLIPSE 100
The output is generated by entering the argument WELSPECS in the RPTSCHED keyword.

ECLIPSE 300
The output is generated by RPTPRINT keyword switches 6 and 7. This allows WELL and
COMPLETION reports to be written which in turn prints out new segment and new completion data in
addition to a multisegment well report.

2. Multisegment wells must use the average density calculation for the hydrostatic head (item 12 of
keyword WELSPECS is automatically be set to AVG).

3. In thermal simulations (Thermal option), heat can be conducted both across and along the pipe wall.
Conduction across the pipe wall (between the well and the formation, or between inner tubing and
outer annulus segments) is governed by the WSEGHEAT keyword. Conduction along the pipe wall,
between each segment and its outlet segment, is determined by the cross-sectional area of the pipe
wall material (item 13) and its thermal conductivity (item 15). Since the top segment has no outlet
segment, its thermal conductivity (item 12 of record 1) is used purely as a default value for the other
segments.

Heat can also be stored in the pipe wall. The amount of energy that it stored within the pipe wall of
each segment depends on the volume of the pipe wall material and its heat capacity. The volume of
pipe wall material is equal to the segment length multiplied by the pipe wall cross-sectional area (item
13). The heat capacity is defined in item 14. The corresponding data items for the top segment (items
10 and 11 of record 1) govern the heat storage in that segment and also act as default values for the
other segments.

Examples

Example 1

To define multilateral well with a main stem and three horizontal lateral branches:

```
WELSEGS
-- Name    Dep 1   Tlen 1   Vol 1   Len&Dep   PresDrop   FlowModel
PROD   7010     0.0       1*      INC       HF-   /
-- First   Last   Branch   Outlet   Length   Depth   Diam    Rough   Area   Vol
-- Seg     Seg    Num      Seg               Change
-- Main Stem
  2       12     1        1        20.0     20.0    0.3    1.0E-3   /
-- Top Branch
  13      13     2        2        50.0     0.0     0.3    1.0E-3   /
  14      17     2        13       100.0    0.0     0.3    1.0E-3   /
-- Middle Branch
  18      18     3        9        50.0     0.0     0.3    1.0E-3   /
  19      22     3        18       100.0    0.0     0.3    1.0E-3   /
-- Bottom Branch
  23      23     4        12       50.0     0.0     0.3    1.0E-3   /
  24      27     4        23       100.0    0.0     0.3    1.0E-3   /
/```

Keywords

2555
Example 2

To change its segments to use the drift flux slip model at a later stage in the run, it is necessary to reset the well’s default slip model to DF and then use WSEGFMOD to change its segments to use the drift flux model:

```
WELSEGS
   -- Name    Dep 1   Tlen 1   Vol 1   Len&Dep   PresDrop   FlowModel
PROD     3*                       INC       HF-         DF  /
/
WSEGFMOD
   -- Well    Segments     Flow   Local   Prof   Prof   Prof
   -- Name    From  To     Model  Slip     A     B     v mult
PROD     2    27      DF    /
/
```
WELSONIN  Minimum oil saturation for automatic opening

The keyword should be followed by a single real number which represents the minimum oil saturation in a grid block necessary to allow a well connection to be opened automatically in it. The number must be followed by a slash (/).

A connection on automatic opening (see item 6 of keyword COMPDAT) is not opened during a workover if the grid block has an oil saturation below this value. If a well is opened from the drilling queue (keyword QDRILL or WDRILPRI) or on automatic opening (see item 2 in keyword WCONPROD), any connection in a grid block with an oil saturation below this value is not opened, but will be placed under automatic opening instead.

If the COMPLUMP keyword is used to lump connections together into completions, the minimum oil saturation value applies to the average oil saturation in the grid blocks connecting with a completion, weighted in proportion to their connection transmissibility factors. If this average saturation exceeds the minimum value, all the connections in the completion are opened. If this average saturation is less than the minimum value, none of the connections in the completion are allowed to open.

This limit does not prevent connections being opened manually (for example using keyword COMPDAT).

If this keyword is omitted, automatic opening of connections takes place regardless of the grid block oil saturation.

Example

```plaintext
WELSONIN
  0.2 /
```
WELSPECL General specification data for wells in local grids

WELSPECL must be used in place of WELSPECS to set the general specification data for wells in local refined grids. The keyword data is similar to that for WELSPECS, except there is an additional item at position 3 which gives the name of the local grid refinement in which the well is located. The connection data for these wells must be specified using the keyword COMPDATL instead of COMPDAT.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated early with a slash (/); the remaining items assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Well name
   (Up to 8 characters)

2. Name of the group to which the well belongs
   (Up to 8 characters)

   **ECLIPSE 100**
   The name FIELD cannot be used as a group name here. Also, node-groups defined with keyword GRUPTREE cannot contain wells.

   A well can be reassigned to a different group during the simulation, if required, by reentering its WELSPECL data with the new group name.

   **ECLIPSE 300**
   DEFAULT: FIELD

3. Name of the local refined grid in which the well is located

   **ECLIPSE 100**
   If the well is completed in an amalgamation of local grids (see keyword AMALGAM), the name of any local grid in the amalgamation may be used here. Only one record is needed per well, even for wells which are completed in more than one local grid.

4. I - location of well head, in the coordinate system of the local grid

5. J - location of well head, in the coordinate system of the local grid

   For horizontal wells, items 4 and 5 define the location of the heel of the well, which ECLIPSE may use to determine the order of the connections (see the COMPORD keyword). If all connections are vertical, items 4 and 5 are for reporting purposes only; the well can still connect with grid blocks having different I- and J-locations. See keyword COMPDATL.

6. Reference depth for bottom hole pressure

   This should be situated in the vicinity of the perforations. A recommended location is the top-most perforation of the well.

   If this quantity is defaulted or given a negative value, the reference depth is set equal to the center depth of the grid block containing the top-most connection in the well.

   In multisegment wells (See "Multisegment wells" in the ECLIPSE Technical Description) the bottom hole reference depth is set equal to the depth of the well’s top segment node entered in record 1 of keyword WELSEGS.

   **UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT–M)

---

**Keywords**

2558
7. Preferred phase for the well

OIL The well is primarily an oil well
WATER The well is primarily a water well
GAS The well is primarily a gas well
LIQ May be used to output the oil + water phase P.I.

This information is used to define the productivity/injectivity index of the well, or its potential flow rate (see item 8). It can also be used in the group/field production rate control option (see item 7 in keyword GCONPROD).

8. Drainage radius for productivity/injectivity index calculation

The instantaneous productivity/injectivity index is printed in the well reports. It is calculated from a steady-state relationship, which is described in "The productivity index" in the ECLIPSE Technical Description.

If a zero value is specified, the pressure equivalent radius of the (local) grid blocks containing the well connections is used. The productivity/injectivity index is then equal to the sum over the connections of the product of the connection factor and the local mobility of the preferred phase specified in item 7.

ECLIPSE 100

If a negative value is specified, the well’s potential flow rate of the preferred phase is printed in the well reports instead of the productivity/injectivity index. The well’s potential rate is the flow rate it would achieve in the absence of any rate constraint, limited only by its BHP and/or THP constraint, at the current grid block conditions - see "Well potentials" in the ECLIPSE Technical Description.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
DEFAULT: 0.0

9. Flag for use of a special inflow equation to model the flow of gas between the completed grid blocks and the well completions

STD or NO The standard inflow equation is used
R-G or YES The Russell-Goodrich equation
P-P The dry gas pseudo-pressure equation
GPP The generalized pseudo-pressure equation

(ECLIPSE 100 only)

The Russell-Goodrich and dry gas pseudo-pressure equations just alter the free gas mobility to take account of its pressure dependence between the grid block pressure and the connection pressure. The generalized pseudo-pressure equation alters both the gas and oil mobilities, and takes account also of the effects of condensate dropout. See "The phase mobilities" in the ECLIPSE Technical Description for a description of these options.

(ECLIPSE 300 only)

The generalized pseudo-pressure method may be activated in ECLIPSE 300 for all wells by entering the keyword PSEUPRES early in the SCHEDULE section. This will override item 9 above.

DEFAULT: STD
10. Instructions for automatic shut-in

**STOP** Stop well above formation (allowing crossflow)

**SHUT** Isolate well from the formation

The well can be automatically shut or stopped if a well or group economic limit, or a group flow limit, is violated.

DEFAULT: **SHUT**

11. Crossflow ability flag

**YES** Crossflow allowed in the well

**NO** Crossflow not allowed; the connections act as one-way valves, preventing flow if there is a reverse drawdown

DEFAULT: **YES**

12. Pressure table number for wellbore fluid properties

(This governs the calculation of wellbore hydrostatic head at formation level, and the relation between surface rates and reservoir fluid volume rate.)

**ECLIPSE 100 only** If a zero value is entered, the pressure table number is set equal to the PVT region number of the lowest grid block in which the well is completed.

**ECLIPSE 300 only** If a zero value is entered, the pressure table number is set equal to the PVT region number of first grid block in which the well is completed.

DEFAULT: 0

13. Type of density calculation for the wellbore hydrostatic head

**SEG** Segmented density calculation.

The flowing mixture density is calculated between each adjacent pair of connections, and depends on the relative quantities of the phases flowing in these particular sections of the wellbore. This calculation is more accurate than the other choice if different mixtures of fluids enter the well through each connection, but it is treated *explicitly* in the numerical scheme.

**AVG** Averaged density calculation.

The flowing mixture density is treated as uniform throughout the wellbore at formation level, and depends on the total inflow rates of each phase and the well’s bottom hole pressure. This calculation is performed *implicitly* in the numerical scheme.

It is usually best to use **SEG**, for its greater accuracy, and change to **AVG** only if the explicit calculation causes stability problems.

The hydrostatic heads calculated using segmented densities in the wellbore are damped to reduce the likelihood of stability problems by combining values from the previous timestep with those of the current timestep in a weighted average.

**ECLIPSE 100**

- The damping is only applied to stopped wells or wells with a zero rate target, and the weighting used in the damping is calculated internally.
• The damping is applied to both open and stopped wells, and the weighting used in the damping can be adjusted using item 44 in the OPTIONS3 keyword.

**CAUTION:** An error in the damping algorithm was fixed in the 2010.1 version of the simulators. This may change simulation results. To revert to the previous behavior, use item 145 in the OPTIONS keyword in ECLIPSE 100, and item 213 of the OPTIONS3 keyword in ECLIPSE 300.

This item is ignored for multisegment wells (see "Multisegment wells" in the ECLIPSE Technical Description). In these wells the average mixture density is calculated implicitly within each segment.

**DEFAULT:** SEG

**ECLIPSE 300**

The default density calculation may be changed using item 65 in the OPTIONS3 keyword.

14. Fluids-in-place region number supplying the reservoir conditions used in calculating the well’s reservoir fluid volume rate.

(The reservoir fluid volume rate is based on the volume that the produced or injected mixture would occupy at the average conditions - hydrocarbon pressure, $R_s$, $R_v$ - within the specified fluids-in-place region.)

If a zero value is entered, the average hydrocarbon conditions in the global grid system is used.

If a negative value is entered, the region number is set equal to the fluids-in-place region number of the lowest grid block in which the well is completed.

If a positive value is entered, this is interpreted as the required fluids-in-place region number. A warning message is printed if the well is not completed in a grid block within that region.

**DEFAULT:** 0

15. Reserved for use by FrontSim.

16. Reserved for use by FrontSim.

**ECLIPSE 300**

17. Specify the type of well model:

   **STD** The standard well model.

   **HMIW** High mobility injection wells. An average completion pressure is considered, this is to overcome convergence problems when the completion mobilities in the injection wells are high. See "High mobility injection wells" in the ECLIPSE Technical Description.

   **DEFAULT:** STD

**ECLIPSE 100**

18. Polymer mixing table number for wellbore fluid properties.

   This determines how polymer affects wellbore fluid viscosity for friction wells and multisegment wells. It has no effect for standard wells and no effect if POLYMER has not been activated.

   If a zero value is entered, the polymer mixing table number is set equal to the PLMIXNUM table number of the lowest grid block in which the well is completed.

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.
**Note:** Remember to specify the well separator conditions with `WSEPCOND` when using `WELSPECL`.

**Note:** In a parallel run, if an LGR is switched off, a well cannot be defined within an LGR.

### Examples

#### Example 1

```
WELSPECL
PROD1  G1  LGR1  6  8  6000  GAS  0.0  Y  STOP  N  2  AVG  1 /
PROD2  G1  LGR2  5  9  6050  OIL  5000 1*  STOP  2*  AVG  2 /
PROD3  G2  LGR3  4  7  6050  OIL  /
```

#### Example 2

```
WELSPECL
PROD1  G1  LGR1  6  8  6000  OIL  6*  1 /
```

**Keywords**

2562
**WELSPECS**  **General specification data for wells**

The keyword introduces a new well, defining its name, the position of the wellhead, its bottom hole reference depth and other specification data.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated early with a slash (/); the remaining items will assume their default values. Default values can also be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. **Well name**  
   (Up to 8 characters)

2. **Name of the group to which the well belongs**  
   (Up to 8 characters)

   **ECLIPSE 100**  
   The name `FIELD` cannot be used as a group name here. Also, node-groups defined with keyword `GRUPTREE` cannot contain wells. **It is strongly recommended that this item is not defaulted in ECLIPSE 100.**

   A well can be re-assigned to a different group during the simulation, if required, by re-entering its WELSPECS data with the new group name.

   **ECLIPSE 300**  
   **DEFAULT:** `FIELD`

3. **I - location of well head or heel**

4. **J - location of well head or heel**

   For horizontal wells, items 3 and 4 define the location of the heel of the well, which ECLIPSE may use to determine the order of the connections (see the COMPORD keyword). If all connections are vertical, items 3 and 4 are used for reporting purposes only; the well can still connect with grid blocks having different I- and J-locations.

5. **Reference depth for bottom hole pressure**

   This should be situated in the vicinity of the perforations. A recommended location is the topmost perforation of the well.

   In multisegment wells the bottom hole reference depth is set equal to the depth of the well’s top segment node entered in record 1 of keyword WELSEGS.

   **DEFAULT:** If this quantity is defaulted or given a negative value, the reference depth is set equal to the center depth of the grid block containing the first connection in the well as defined by COMPORD.

   **ECLIPSE 100**  
   If the grid has one or more cells above sea level, then a negative value is taken as a reference depth.

   **UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

6. **Preferred phase for the well**

   **OIL**  
   The well is primarily an oil well

   **WATER**  
   The well is primarily a water well
GAS    The well is primarily a gas well

LIQ    May be used to output the oil + water phase P.I.

This information is used to determine the worst offending well or connection for closure when a group production rate limit is exceeded in GCONPROD (see item 7) and GCONPRI (see item 3).

ECLIPSE 100

This information is also used to define the productivity/injectivity index of the well, or its potential flow rate (see item 7).

7. Drainage radius for productivity/injectivity index calculation.

The instantaneous productivity/injectivity index is printed in the well reports. It is calculated from a steady-state relationship, which is described in "The productivity index" in the ECLIPSE Technical Description.

If a zero value is specified, the pressure equivalent radius of the grid blocks containing the well connections is used. The productivity/injectivity index is then equal to the sum over the connections of the product of the connection factor and the local mobility of the preferred phase specified in item 6.

ECLIPSE 100

If a negative value is specified, the well’s potential flow rate of the preferred phase is printed in the well reports instead of the productivity/injectivity index. The well’s potential rate is the flow rate it would achieve in the absence of any rate constraint, limited only by its BHP and/or THP constraint, at the current grid block conditions - see "Well potentials" in the ECLIPSE Technical Description.

(When running the Gas Field Operations Model, the well’s gas delivery capacity is printed instead, for a production well.)

UNITs: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: 0.0

8. Flag for use of a special inflow equation to model the flow of gas between the completed grid blocks and the well completions

    STD or NO    The standard inflow equation will be used

ECLIPSE 100 only

    R–G or YES   The Russell-Goodrich equation

ECLIPSE 100 only

    P–P        The dry gas pseudo-pressure equation

    (This option is not available in gas condensate runs.)

    GPP        The generalized pseudo-pressure equation.

    (This option may be used by gas condensate producers. See also keyword PICOND.)

The Russell-Goodrich and dry gas pseudo-pressure equations just alter the free gas mobility to take account of its pressure dependence between the grid block pressure and the connection pressure. The generalized pseudo-pressure equation alters both the gas and oil mobilities, and takes account also of the effects of condensate dropout. See "Special inflow equations" in the ECLIPSE Technical Description for a description of these options.

ECLIPSE 300 only

The generalized pseudo-pressure method may be activated in ECLIPSE 300 for all wells by entering the keyword PSEUPRES early in the SCHEDULE section. This will override item 8 above.

DEFAULT: STD
9. Instructions for automatic shut-in

STOP  Stop well above formation (allowing crossflow)

SHUT  Isolate well from the formation.

The well can be automatically shut or stopped if a well or group economic limit, or a group flow limit, is violated.

DEFAULT: SHUT

10. Crossflow ability flag

YES  Crossflow allowed in the well

NO  Crossflow not allowed; the connections act as one-way valves, preventing flow if there is a reverse drawdown

DEFAULT: YES

11. Pressure table number for wellbore fluid properties

(This governs the calculation of wellbore hydrostatic head at formation level, and the relation between surface rates and reservoir fluid volume rate)

ECLIPSE 100 only  If a zero value is entered, the pressure table number is set equal to the PVT region number of the lowest grid block in which the well is completed.

ECLIPSE 300 only  If a zero value is entered, the pressure table number is set equal to the PVT region number of first grid block in which the well is completed.

DEFAULT: 0

12. Type of density calculation for the wellbore hydrostatic head

SEG  Segmented density calculation.

The flowing mixture density is calculated between each adjacent pair of connections, and depends on the relative quantities of the phases flowing in these particular sections of the wellbore. This calculation is more accurate than the other choice if different mixtures of fluids enter the well through each connection, but it is treated explicitly in the numerical scheme.

AVG  Averaged density calculation.

The flowing mixture density is treated as uniform throughout the wellbore at formation level, and depends on the total inflow rates of each phase and the well’s bottom hole pressure. This calculation is performed implicitly in the numerical scheme.

It is usually best to use SE, for its greater accuracy, and change to AVG only if the explicit calculation causes stability problems.

The hydrostatic heads calculated using segmented densities in the wellbore are damped to reduce the likelihood of stability problems by combining values from the previous timestep with those of the current timestep in a weighted average.

ECLIPSE 100

•  The damping is only applied to stopped wells or wells with a zero rate target, and the weighting used in the damping is calculated internally.

ECLIPSE 300
• The damping is applied to both open and stopped wells, and the weighting used in the damping can be adjusted using item 44 in the OPTIONS3 keyword.

**Note:** An error in the damping algorithm has been identified and fixed in the 2010.1 version of the simulators. This fix has not been made to the default behavior as it may change simulation results. To use the correct damping, use item 145 in the OPTIONS keyword in ECLIPSE 100, and item 213 of the OPTIONS3 keyword in ECLIPSE 300.

This item is ignored for multisegment wells (see "Multisegment wells" in the ECLIPSE Technical Description). In these wells the average mixture density is calculated implicitly within each segment.

**DEFAULT:** SEG

**ECLIPSE 300**

The default density calculation may be changed using item 65 in the OPTIONS3 keyword

13. Fluids-in-place region number supplying the reservoir conditions used in calculating the well’s reservoir fluid volume rate.

(The reservoir fluid volume rate is based on the volume that the produced or injected mixture would occupy at the average conditions - hydrocarbon pressure, \(R_s\), \(R_v\) - within the specified fluids-in-place region).

If a zero value is entered, the average hydrocarbon conditions in the field is used.

If a negative value is entered, the region number is set equal to the fluids-in-place region number of the lowest grid block in which the well is completed.

If a positive value is entered, it is interpreted as the required fluids-in-place region number. A warning message is printed if the well is not completed in a grid block within that region.

**DEFAULT:** 0

14. Reserved for use by FrontSim.

15. Reserved for use by FrontSim.

**ECLIPSE 300**

16. Specify the type of well model:

```
STD  The standard well model.

HMIW  High mobility injection wells. An average completion pressure is considered, this is to overcome convergence problems when the completion mobilities in the injection wells are high. See "High mobility injection wells" in the ECLIPSE Technical Description
```

**DEFAULT:** STD

**ECLIPSE 100**

17. Polymer mixing table number for wellbore fluid properties.

This determines how polymer affects wellbore fluid viscosity for friction wells and multisegment wells. It has no effect for standard wells and no effect if POLYMER has not been activated.

If a zero value is entered, the polymer mixing table number is set equal to the PLMIXNUM table number of the lowest grid block in which the well is completed.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.
**Note:** Remember to specify the well separator conditions with `WSEPCOND` when using `WELSPECS`.

## Examples

### Example 1

```
ECLIPSE 100
WELSPECS
  PROD1  G1  6 8 6000  GAS  0.0  Y  STOP  N  2  AVG  1 /
  PROD2  G1  5 9 6050  OIL  5000 1*  STOP  2*  AVG  2 /
  PROD3  G2  4 7 6050  OIL /
```

### Example 2

```
ECLIPSE 300
WELSPECS
  PROD1  G1  6 8 6000  OIL  6*  1 /
```

---

**Keywords**

2567
Resets a well operating target or limit

This keyword can be used to reset a target or limit value for a well, without having to re-specify all the other quantities required by the control keywords `WCONPROD` or `WCONINJE`. These other quantities are left unchanged, including the open/shut status of the well. The well control data must initially have been fully specified using `WCONPROD` or `WCONINJE`.

If the well has been declared a history matching well (see keywords `WCONHIST` and `WCONINJH`) the `WELTARG` keyword may be used to modify its BHP limit, VFP table number, and artificial lift quantity. The other quantities should not be modified with this keyword.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. **Well name, well name template, well list or well list template**
   - A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the *ECLIPSE Technical Description* for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword `WLIST`.

2. **Definition of the control or constraint quantity to be changed**
   - `ORAT`  Oil rate
   - `WRAT`  Water rate
   - `GRAT`  Gas rate
   - `LRAT`  Liquid rate
   - `CRAT`  Linearly combined rate (coefficients must be set with `LINCOM`)
   - `RESV`  Reservoir fluid volume rate
   - `BHP`  Bottom hole pressure
   - `THP`  Tubing head pressure (A non-zero VFP table number must first have been set)
   - `VFP`  VFP table number
   - `LIFT`  Artificial lift quantity (A non-zero VFP table number must first have been set)
   - `GUID`  Guide rate (For group controlled wells)

   *ECLIPSE 300 only*  
   - `WGRA`  Wet gas rate

   *ECLIPSE 300 only*  
   - `NGL`  NGL rate

   *ECLIPSE 300 only*  
   - `CVAL`  Calorific production rate

   *ECLIPSE 300 only*  
   - `REIN`  Reinjection fraction

   *ECLIPSE 300 only*  
   - `STRA`  Steam rate (Available only with the Thermal option)
**ECLIPSE 300 only**

SATP  Saturation pressure offset. (Available only with the Thermal option)

**ECLIPSE 300 only**

SATT  Saturation temperature offset (Available only with the Thermal option)

3. New value of this quantity.

   **UNITS:** The units depend upon the quantity chosen. The appropriate units are listed in the specification for keywords \texttt{WCONPROD} and \texttt{WCONINJE}.

   **DEFAULT:** If the quantity in item 2 is \texttt{VFP}, \texttt{LIFT} or \texttt{GUID}, then the value here may not be defaulted. For all other quantities, a defaulted value here will result in ECLIPSE using the corresponding flowing value for the well at the end of the previous timestep as the target or limit for the control or constraint in item 2.

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also keyword \texttt{WTMULT}.

**Example**

```
WELTARG
  PROD1 ORAT 1500 /
  PROD2 BHP 3000 /
/
```
WFOAM

Sets foam concentration for injection wells

This keyword is used to specify the concentration of foam in the injection stream of each well, when the Foam Model is enabled. If the WFOAM keyword does not appear in the injection schedule, then concentration values of zero are assumed.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. The concentration of foam in the injection stream for the well. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

   If item 1 of FOAMOPTS is set to ‘GAS’, then the well should be a gas injector, and the concentration units are:

   UNITS: kg/sm$^3$ (METRIC), lb/Mscf (FIELD), gm/scc (LAB)

   If item 1 of FOAMOPTS is set to ‘WATER’, then the well should be a water injector and the concentration units are:

   UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scf (LAB)

End data record with a slash (/).

End the set of records with a blank record, containing just a slash (/).

See also keyword WCONINJE.

Example

```
WFOAM
  INJF1  0.04  /
  INJF2  0.01  /
```

Keywords

2570
WFRICSEG  Converts friction well WFRICTN data into a multisegment well

This keyword may be used to define both the segment structure and the connection locations of a multisegment well, using the data from the WFRICTN keyword. It replaces both the WELSEGS keyword and the COMPSEGS keyword. It thus provides an easy way of converting a friction well into a multisegment well, simply by changing the keyword name to WFRICSEG. (See "Converting friction wells to multisegment wells" in the ECLIPSE Technical Description.)

Friction wells that are situated in local grids and use the WFRICTNL keyword should be converted by changing the keyword to WFRICSGL.

Note that friction wells and multisegment wells cannot coexist in the same run. All the friction wells must be converted if this facility is used. Also the FRICTION keyword must be removed from the RUNSPEC section, replacing it with the WSEGDIMS keyword with appropriate dimensions.

ECLIPSE creates the segment structure such that there is one segment per grid block connection, plus an additional segment at the well’s bottom hole reference point. The segments are distributed so that the nodal point of each segment is located at the center of the corresponding grid block connection. The segment lengths are set equal to the tubing length between adjacent connections. The depths of the segment nodes are set equal to the center depths of the connecting grid blocks, as there is no data in WFRICTN that defines the depth trajectory of the well bore. ECLIPSE outputs the resulting WELSEGS and COMPSEGS data to the Print file in the appropriate keyword format. The keyword data may then be copied, if desired, to replace the WFRICSEG keyword in the data file where it can be edited for additional flexibility. It is advisable to readjust the segment depths to correspond to the actual well trajectory, as setting them equal to the grid block center depths produces spurious undulations if the grid is not aligned with the well.

When the WFRICTN data represents a multilateral well, ECLIPSE creates the branches and numbers them. However, ECLIPSE is unable to recognize whether the original main stem continues past a branch junction. Instead, two new branches are created at each branch junction, one of these being the continuation of the main stem. It is important to allow for this when setting the maximum number of branches in WSEGDIMS. For example, a well consisting of a main stem and a single lateral branch will be converted to have three branches: the upper part of the main stem, the lateral branch and the section of the main stem below the branch junction. You can reduce the number of branches, if desired, by copying the output WELSEGS and COMPSEGS data and editing it to renumber all the main stem sections as branch 1 (don’t forget to edit the branch numbers in COMPSEGS accordingly).

The WFRICSEG keyword data items are identical to those of the WFRICTN keyword.
WFRICSGL  Converts friction well WFRICTNL data into a multisegment well

This keyword may be used to define both the segment structure and the connection locations of a multisegment well situated in a local grid (or an amalgamated set of local grids), using the data from the WFRICTNL keyword. It replaces both the WELSEGS keyword and the COMPSEGL keyword. It thus provides an easy way of converting a friction well into a multisegment well, simply by changing the keyword name to WFRICSGL. (See "Converting friction wells to multisegment wells" in the ECLIPSE Technical Description.)

Note that friction wells and multisegment wells cannot coexist in the same run. All the friction wells must be converted if this facility is used. Also the FRICTION keyword must be removed from the RUNSPEC section, replacing it with the WSEGDIMS keyword with appropriate dimensions.

ECLIPSE creates the segment structure such that there is one segment per local grid block connection, plus an additional segment at the well’s bottom hole reference point. The segments are distributed so that the nodal point of each segment is located at the center of the corresponding grid block connection. The segment lengths are set equal to the tubing length between adjacent connections. The depths of the segment nodes are set equal to the center depths of the connecting grid blocks, as there is no data in WFRICTNL that defines the depth trajectory of the well bore. ECLIPSE outputs the resulting WELSEGS and COMPSEGL data to the Print file in the appropriate keyword format. The keyword data may then be copied, if desired, to replace the WFRICSGL keyword in the data file where it can be edited for additional flexibility. It is advisable to readjust the segment depths to correspond to the actual well trajectory, as setting them equal to the grid block center depths produces spurious undulations if the grid is not aligned with the well.

When the WFRICTNL data represents a multilateral well, ECLIPSE creates the branches and numbers them. However, ECLIPSE is unable to recognize whether the original main stem continues past a branch junction. Instead, two new branches are created at each branch junction, one of these being the continuation of the main stem. It is important to allow for this when setting the maximum number of branches in WSEGDIMS. For example, a well consisting of a main stem and a single lateral branch will be converted to have three branches: the upper part of the main stem, the lateral branch and the section of the main stem below the branch junction. You can reduce the number of branches, if desired, by copying the output WELSEGS and COMPSEGL data and editing it to renumber all the main stem sections as branch 1 (don’t forget to edit the branch numbers in COMPSEGL accordingly).

The WFRICSGL keyword data items are identical to those of the WFRICTNL keyword.
WFRICTN  Designates a well as a friction well

This keyword is used to designate a well to be a friction well, for which the frictional pressure losses between the connections and the well’s bottom hole reference point are calculated and included in the connection head terms. This facility is primarily intended for use with horizontal and multilateral wells, in which frictional pressure losses may be significant over the horizontal section of the wellbore and in the branches. (See "Wellbore friction option" in the ECLIPSE Technical Description.)

This keyword can only be used on one well at a time, so the keyword must be specified more than once if there is more than one friction well. If the keyword is repeated for a particular well (for example to add a new branch or new connections), the data for the entire well must be supplied each time the keyword is used.

If the well is completed in one or more local grids that have been amalgamated (see "Local grid refinement and coarsening" in the ECLIPSE Technical Description), the keyword WFRICTNL must be used instead of WFRICTN. Either keyword may be used for wells completed in single unamalgamated local grids.

The keyword is followed by a number of data records to describe the properties and geometry of the (possibly multibranched) well. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

The data items in each record are:

**Record 1**

The first record identifies the well and sets the tubing diameter, roughness and flow scaling factor, which are used in the friction pressure loss calculation.

1. Name of the well
2. Tubing internal diameter
   
   This quantity is used for calculating the frictional pressure drop between the connections and the well’s bottom hole reference point. It need not be the same as the wellbore diameter set in COMPDAT item 9, which is used to calculate the connection transmissibility factors.

   Different internal diameters may be set for individual connections, with item 8 in subsequent records.

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
3. The effective absolute roughness of the tubing
   
   This quantity is used to calculate the Fanning friction factor.

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
4. Flow scaling factor (dimensionless)

   The volumetric flow rate in the wellbore is multiplied by the scaling factor when the friction pressure drop is calculated. This quantity is useful if the well is being simulated in a half or quarter grid, taking advantage of symmetry. In this case, the well’s flow rate will only be a half or quarter of the true flow rate. The scaling factor can then be used to multiply the connection flow rates by 2 or 4, so that the true flow rate is used in the friction calculation.

   DEFAULT: 1.0

End the data record with a slash (/).
Each subsequent record

The subsequent records set the start and end of each connection’s perforations in terms of distance along the tubing from the well’s bottom hole reference point. These distances are required by the frictional pressure drop calculation, which treats each connection’s flow as entering the tubing half way between the start and end of the perforations in the grid block. The bottom hole reference point should be downstream for a producer (upstream for an injector) of all the perforations. A suitable position is at the start of the perforations (the end nearest the wellhead). The order in which these records are entered is used to determine the structure of the well (see “Multilateral wells” in the ECLIPSE Technical Description). For standard horizontal wells, the connections should be entered in order of increasing distance from the bottom hole reference point, that is from the heel out to the toe.

The connections may either be specified individually, or in one or more “ranges”. A range is a contiguous row or column of grid blocks that are fully penetrated by the well. The well is assumed to penetrate a range perpendicularly through the center of each grid block in the specified direction. The distance along the tubing to the start and end of each connection in the range is determined by summing the DX, DY or DZ thicknesses of the grid blocks in the range. All connections in the well must have their tubing distances set, either individually or within a range, working from the connection nearest the well’s bottom hole reference point out to the furthest connection of the well.

For a standard horizontal well, each connection should be referenced by this keyword just once, either individually or as part of a range. For multilateral wells, the topology of the branches is defined by referencing specific connections again at each branch point. If a connection is referenced again in WFRICTN after its first entry, either singly or as the first connection of a range, then it is taken to indicate that the subsequent connections form a branch that joins the well at this location. The connections in this branch should be ordered from the join out towards its toe. The properties of branch point connections are defined by items 4 - 8 of the record in which they are first referenced; these items are ignored when a connection is referenced again. All connections must of course have been defined initially in keyword COMPDAT. Note that there must therefore be a connection defined at each branch point. However, if the well is not actually perforated within these grid blocks the connection should be declared as SHUT in keyword COMPDAT. The maximum number of branches emanating from any one connection must be entered in keyword FRICTION in the RUNSPEC section. See “Multilateral wells” in the ECLIPSE Technical Description for further details and examples.

1. I - location of the individual connection, or the connection at the start of the range (nearest the well’s bottom hole reference point)
2. J - location of the individual connection, or the connection at the start of the range (nearest the well’s bottom hole reference point)
3. K - location of the individual connection, or the connection at the start of the range (nearest the well’s bottom hole reference point)
4. Distance down the tubing from the well’s bottom hole reference point to the start of the perforations in this grid block
   This item must not be negative.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   DEFAULT: Zero, for the first of these records, or the distance to the end of the previous connection or range, for subsequent records
5. Distance down the tubing from the well’s bottom hole reference point to the end of the perforations in this grid block
   This item, if set, must be greater than item 4.
UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: Value of item 4 + thickness of the grid block in the direction of penetration

6. Direction of penetration through the grid block or the range:
   X or I  for horizontal penetration in the x-direction
   Y or J  for horizontal penetration in the y-direction
   Z or K  for vertical penetration

   This item is used for calculating the distance to the end of the perforations in the connection (or the connections in the range), where it defines which component of the grid block size to add (DX, DY or DZ). The item must be set if item 5 is defaulted, or if a range is specified in item 7. Otherwise it may be defaulted as it is not required.

7. Grid block I, J, or K coordinate at the end of the range, depending on the direction of penetration set in item 6. (For example the I-coordinate of the grid block at the end of the range, if the well penetrates in the I-direction.)

   This may be greater than, or less than, the corresponding grid block coordinate in item 1, 2 or 3, depending on whether the well penetrates out towards the positive or negative I, J or K direction.

   If this item is set, the tubing lengths are calculated for all the well’s connections that lie within the range. Should the range pass through an inactive grid block ECLIPSE prints a warning message and assume a zero thickness for that cell. If this is not appropriate, the range should be split and the thickness of the inactive grid block accounted for by explicitly defining the tubing lengths in items 4 and 5.

   If this item is defaulted, this record sets the tubing lengths for the single connection identified by items 1, 2 and 3.

8. Tubing internal diameter for this connection, or range of connections.

   A value entered here is used instead of the value entered in item 2 of "Record 1". This allows, for example, branches in multilateral wells to have different diameters.

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

   DEFAULT: The value entered in item 2 of Record 1.

End the data record with a slash (/).

End the set of data records with a slash.

Notes

• Friction wells must use the segmented density calculation for the hydrostatic head (SEG in item 12 of keyword WELSPECS).

• Multilateral well data can be complicated to enter. It is advisable to check your data by examining the output generated by RPTSCHED keyword switch 14 (mnemonic WELSPECS).

• Friction wells must not have connections in coarsened grid cells.
Examples

A well called HORIZ with 6 connections.

Example 1

This example sets the tubing lengths manually for each connection of the well 'HORIZ'

\[
\begin{array}{lll}
\text{WFRICTN} & \text{-- Name} & \text{Diam} \quad \text{Roughness} \quad \text{Scalefac} \\
HORIZ & 0.25 & 1.0E-3 & 1.0 / \\
\text{-- I J K} & \text{Tlen1} & \text{Tlen2} & \text{Dirn} \quad \text{RangeEnd} \quad \text{Diam} \\
3 & 5 & 6 & 0.0 & 100.0 / \\
4 & 5 & 6 & 100.0 & 200.0 / \\
5 & 5 & 6 & 200.0 & 400.0 / \\
6 & 5 & 6 & 400.0 & 600.0 / \\
7 & 5 & 6 & 600.0 & 700.0 / \\
8 & 5 & 6 & 700.0 & 800.0 / \\
/ \\
\end{array}
\]

Example 2

This example sets the tubing lengths for the same set of well connections grouped into a single range. The well is assumed to penetrate fully the grid blocks I=3-8, J=5, K=6, with the connection at I=3 nearest the wellhead

\[
\begin{array}{lll}
\text{WFRICTN} & \text{-- Name} & \text{Diam} \quad \text{Roughness} \quad \text{Scalefac} \\
HORIZ & 0.25 & 1.0E-3 & 1.0 / \\
\text{-- I J K} & \text{Tlen1} & \text{Tlen2} & \text{Dirn} \quad \text{RangeEnd} \quad \text{Diam} \\
3 & 5 & 6 & 0.0 & 1* & I & 8 / \\
/ \\
\end{array}
\]

Example 3

This example defines a simple multilateral well, having a horizontal main stem (3-8, 5, 6) based on the previous example and a single perpendicular side-branch (5, 5-1, 6) of smaller diameter, joining the main stem at (5, 5, 6). Note that the branch point connection (5, 5, 6) is referenced twice: firstly as part of the main stem range, and secondly as the start of the side-branch range. The properties of the connection are taken from the first reference (that is, I-direction penetration and a diameter of 0.25).

\[
\begin{array}{lll}
\text{WFRICTN} & \text{-- Name} & \text{Diam} \quad \text{Roughness} \quad \text{Scalefac} \\
MULTI & 0.25 & 1.0E-3 & 1.0 / \\
\text{-- I J K} & \text{Tlen1} & \text{Tlen2} & \text{Dirn} \quad \text{RangeEnd} \quad \text{Diam} \\
3 & 5 & 6 & 0.0 & 1* & I & 8 & / \text{main stem (3-8, 5, 6)} \\
5 & 5 & 6 & 2* & & J & 1 & 0.19 & / \text{side branch (5, 5-1, 6)} \\
/ \\
\end{array}
\]

Or, equivalently,

\[
\begin{array}{lll}
\text{WFRICTN} & \text{-- Name} & \text{Diam} \quad \text{Roughness} \quad \text{Scalefac} \\
MULTI & 0.25 & 1.0E-3 & 1.0 / \\
\text{-- I J K} & \text{Tlen1} & \text{Tlen2} & \text{Dirn} \quad \text{RangeEnd} \quad \text{Diameter} \\
3 & 5 & 6 & 0.0 & 1* & I & 8 & / \text{main stem (3-8, 5, 6)} \\
5 & 5 & 6 & & & & & / \text{branch-point (5, 5, 6)} \\
5 & 4 & 6 & 2* & & J & 1 & 0.19 & / \text{side branch (5, 4-1, 6)} \\
/ \\
\end{array}
\]
WFRICTNL Designates a well in a local grid as a friction well

This keyword is used to designate a well in a local grid to be a friction well for the Wellbore Friction option (see "Wellbore friction option" in the ECLIPSE Technical Description). WFRICTNL performs the same role as does WFRICTN, but the local grid name should be specified to allow for wells completed in amalgamated local grids (see keyword AMALGAM).

The frictional pressure losses between the connections and the well’s bottom hole reference point are calculated and included in the connection head terms. This facility is primarily intended for use with horizontal and multilateral wells, in which frictional pressure losses may be significant over the horizontal section of the wellbore and in the branches.

This keyword can only be used on one well at a time, so the keyword must be specified more than once if there is more than one friction well. If the keyword is repeated for a particular well (for example to add a new branch or new connections), the data for the entire well must be supplied each time the keyword is used.

The keyword is followed by a number of data records to describe the properties and geometry of the (possibly multibranched) well. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

The data items in each record are:

Record 1

The first record identifies the well and sets the tubing diameter, roughness and flow scaling factor, which are used in the friction pressure loss calculation.

1. Name of the well
2. Tubing internal diameter
   This quantity is used for calculating the frictional pressure drop between the connections and the well’s bottom hole reference point. It need not be the same as the wellbore diameter set in COMPDATL, which is used to calculate the connection transmissibility factors.
   Different internal diameters may be set for individual connections, with item 9 in subsequent records.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
3. The effective absolute roughness of the tubing
   This quantity is used to calculate the Fanning friction factor.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
4. Flow scaling factor (dimensionless)
   The volumetric flow rate in the wellbore is multiplied by the scaling factor when the friction pressure drop is calculated. This quantity is useful if the well is being simulated in a half or quarter grid, taking advantage of symmetry. In this case, the well’s flow rate will only be a half or quarter of the true flow rate. The scaling factor can then be used to multiply the connection flow rates by 2 or 4, so that the true flow rate is used in the friction calculation.
   DEFAULT: 1.0

End the data record with a slash (/).
Each subsequent record

The subsequent records set the name of the local grid and also the start and end of each connection’s perforations in terms of distance along the tubing from the well’s bottom hole reference point. These distances are required by the frictional pressure drop calculation, which treats each connection’s flow as entering the tubing half way between the start and end of the perforations in the grid block. The bottom hole reference point should be downstream for a producer (upstream for an injector) of all the perforations. A suitable position is at the start of the perforations (the end nearest the wellhead). The order in which these records are entered is used to determine the structure of the well (see “Multilateral wells” in the ECLIPSE Technical Description). For standard horizontal wells, the connections should be entered in order of increasing distance from the bottom hole reference point, that is from the heel out to the toe.

The connections may either be specified individually, or in one or more ranges. A range is a contiguous row or column of grid blocks that are fully penetrated by the well. The well is assumed to penetrate a range perpendicularly through the center of each grid block in the specified direction. The distance along the tubing to the start and end of each connection in the range is determined by summing the DX, DY or DZ thicknesses of the grid blocks in the range. All connections in the well must have their tubing distances set, either individually or within a range, working from the connection nearest the well’s bottom hole reference point out to the furthest connection of the well.

For a standard horizontal well, each connection should be referenced by this keyword just once, either individually or as part of a range. For multilateral wells, the topology of the branches is defined by referencing specific connections again at each branch point. If a connection is referenced again in WFRICNTNL after its first entry, either singly or as the first connection of a range, then it is taken to indicate that the subsequent connections form a branch that joins the well at this location. The connections in this branch should be ordered from the join out towards its toe. The properties of branch point connections are defined by items 5 - 9 of the record in which they are first referenced; these items are ignored when a connection is referenced again. All connections must of course have been defined initially in keyword COMPDATL. Note that there must therefore be a connection defined at each branch point. However, if the well is not actually perforated within these grid blocks the connection should be declared as SHUT in keyword COMPDATL. The maximum number of branches emanating from any one connection must be entered in keyword FRICTION in the RUNSPEC section. See “Multilateral wells” in the ECLIPSE Technical Description for further details and examples.

1. Name of local grid containing the connections or range specified in this record.

   If the local grid containing the well is not amalgamated (using keyword AMALGAM), this item may be defaulted or set to a blank string. The local grid name then defaults to the well’s local grid named in keyword WELSPECL.

   The connection locations in items 2, 3, 4 and 8 refer to their coordinates within this local grid.

2. I - location of the individual connection, or the connection at the start of the range (nearest the well’s bottom hole reference point)

3. J - location of the individual connection, or the connection at the start of the range (nearest the well’s bottom hole reference point)

4. K - location of the individual connection, or the connection at the start of the range (nearest the well’s bottom hole reference point)

5. Distance down the tubing from the well’s bottom hole reference point to the start of the perforations in this grid block.

   This item must not be negative.

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
6. Distance down the tubing from the well’s bottom hole reference point to the end of the perforations in this grid block

This item, if set, must be greater than item 5.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: value of item 5 + thickness of the grid block in the direction of penetration

7. Direction of penetration through the grid block or the range:

X or I for horizontal penetration in the x-direction

Y or J for horizontal penetration in the y-direction

Z or K for vertical penetration

This item is used for calculating the distance to the end of the perforations in the connection (or the connections in the range), where it defines which component of the grid block size to add (DX, DY or DZ). The item must be set if item 6 is defaulted, or if a range is specified in item 8. Otherwise it may be defaulted as it is not required.

8. Grid block I, J, or K coordinate at the end of the range, depending on the direction of penetration set in item 7. (For example the I-coordinate of the grid block at the end of the range, if the well penetrates in the I-direction.)

This may be greater than, or less than, the corresponding grid block coordinate in item 2, 3 or 4, depending on whether the well penetrates out towards the +ve or -ve I, J or K direction.

If this item is set, the tubing lengths are calculated for all the well’s connections that lie within the range. Should the range pass through an inactive grid block ECLIPSE prints a warning message and assume a zero thickness for that cell. If this is not appropriate, the range should be split and the thickness of the inactive grid block accounted for by explicitly defining the tubing lengths in items 5 and 6.

If this item is defaulted, this record will set the tubing lengths for the single connection identified by items 2, 3 and 4.

A range cannot span more than one local grid.

9. Tubing internal diameter for this connection, or range of connections.

A value entered here will be used instead of the value entered in item 2 of Record 1. This allows, for example, branches in multilateral wells to have different diameters.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: The value entered in item 2 of Record 1

End the data record with a slash (/).

End the set of data records with a slash.

Notes

• Friction wells must use the segmented density calculation for the hydrostatic head (SEG in item 13 of keyword WELSPCGL)
• It is still possible to use keyword WFRICTN for wells completed in a single, non-amalgamated local grid.

• Multilateral well data can be complicated to enter. It is advisable to check your data by examining the output generated by RPTSCHED keyword switch 14 (mnemonic WELSPECS).

• Friction wells must not have connections in coarsened grid cells.

Examples

A well called HORIZ with 6 connections in two amalgamated local grids.

Example 1

This example sets the tubing lengths manually for each connection of the well HORIZ which is completed in local grids WEST and EAST.

```plaintext
WFRICTNL
-- Name      Diam      Roughness     Scalefac
HORIZ     0.25      1.0E-3        1.0  /
-- Local Grid -  I J K  Tlen1  Tlen2  Dirn  RangeEnd  Diam
WEST          3 5 6  0.0    100.0  /
WEST          4 5 6  100.0  200.0  /
WEST          5 5 6  200.0  400.0  /
EAST          1 5 3  400.0  600.0  /
EAST          2 5 3  600.0  700.0  /
EAST          3 5 3  700.0  800.0  /
/
```

Example 2

This example sets the tubing lengths for the same set of well connections grouped into two ranges, one for each local grid WEST and EAST. The well is assumed to penetrate fully the grid blocks I=3-5, J=5, K=6 in grid WEST and I=1-3, J=5, K=3 in grid EAST. The connection at I=3 in grid WEST is nearest the wellhead.

```plaintext
WFRICTNL
-- Name       Diam      Roughness      Scalefac
HORIZ      0.25      1.0E-3         1.0  /
-- Local Grid -  I J K  Tlen1  Tlen2   Dirn   RangeEnd   Diam
WEST          3 5 6  0.0    1*       I       5  /
EAST          1 5 3  1*     1*       I       3  /
/
```

The examples of multilateral wells are similarly analogous to those in the WFRICTN keyword in the wellbore friction option, and you are referred there for this feature. The only difference, as above, is the need to specify the name of the local grid.
WGASPROD  Special gas producers for sales gas control

This keyword can be used to declare one or more production wells to be special gas producers, which are opened from the drilling queue in preference to other wells when the sales gas production rate falls below the minimum limit (see item 4 of keyword GCONSALE).

The wells must first be declared as shut producers and given any other operating constraints with keyword WCONPROD, and placed in the drilling queue in the required order with keyword QDRILL. The wells should be placed under GRAT control and given a nominal gas rate in item 6 of keyword WCONPROD. The nominal gas rate is overridden when the wells are opened from the drilling queue, as their gas rate targets are set automatically at multiples of the incremental rate set in item 2 below.

The purpose of special gas producers is to ensure sufficient gas to meet sales gas requirements independently of any oil production target. You must ensure these wells are completed in a region of the field that provides a suitably high GOR. Special gas producers are opened from the drilling queue only when the sales gas rate has fallen below the minimum requirement, and are not opened as part of normal group production control. They are automatically made unavailable for group control (equivalent to specifying NO in item 2 of keyword WGRUPCON), so that their production will not be reduced if a superior group comes under oil rate control. Any limit on the minimum oil saturation necessary for a well to be opened (set with keyword WELSOMIN) is ignored for these wells.

The keyword is followed by any number of records, each containing the following items of data and terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Incremental gas production rate

   The well’s target gas production rate is increased by this amount each time the sales gas rate of a superior group falls below the minimum limit, until either it reaches the maximum number of increments set in item 3 or it becomes constrained by another limit set in WCONPROD.

   This item must be set to a number greater than zero.

   UNITS: sm³/day (METRIC), Mcf/day (FIELD), scc/hr (LAB), sm³/day (PVT-M)

3. Maximum permitted number of increments

   Once this number of increments have been applied to the well, no more increments can be applied and the drilling queue is searched for another special gas producer to open. Thus, the eventual gas production rate is limited to the incremental rate times the maximum number of increments.

   A zero value turns the well back to an ordinary producer.

Example

Define the wells as shut producers and give them any other operating constraints. Place them under gas rate control with a nominal rate limit:

Keywords

2581
Place the special gas producers in the drilling queue along with any other wells:

```plaintext
QDRILL
    PROD4 PROD5 PROD6 GASWEL1 GASWEL2 /
```

Declare the wells as special gas producers and set their incremental rates and maximum number of increments:

```plaintext
WGASPROD
    GASWEL1 3000 4 /
    GASWEL2 4000 2 /
```
**WGMEM**

**Write geomechanics mechanical earth model (MEM) output data**

This keyword is used to write out ECLIPSE 300 Geomechanics data along specified well surveys at specified times of a simulation. Data to be written includes survey properties, mechanical properties such as Young’s modulus, Poisson’s ratio, Biot’s constant, Initial Cohesion, and stresses, pressures as well as wellbore phase flow rates with time.

This keyword is only available to Schlumberger engineers and can only be accessed if the SMA license is obtained.
Sets data for GOR penalty option

The GOR penalty option provides a means of controlling the oil production rate target of individual wells for each month, as a function of their previous month’s average gas-oil ratio. The option is specifically intended to model a production regulation imposed by regulatory bodies in certain countries.

When this facility is activated for individual wells with the \texttt{WGORPEN} keyword, the wells’ oil production rate target/limit for each calendar month is automatically set to be

\[
\text{Oil Rate} = \frac{(\text{Base GOR}) \cdot (\text{Maximum Rate Limit})}{\text{Average GOR over previous month}} \quad \text{Eq. 3.231}
\]

subject to the condition that it does not exceed the maximum rate limit. The base GOR and the maximum rate limit are parameters you enter.

While this facility is in use, the timesteps are automatically restricted to end at the end of each calendar month (although reports will only be printed at the report times entered with \texttt{DATES, TIME} or \texttt{TSTEP}). At the end of each month, the average gas-oil ratio of each well’s production over the month is determined. At the beginning of the next timestep, the oil production rate target/limit for the new month is set according to the formula above, for each well using the facility.

The calculated oil rate target/limit overwrites any value for this quantity that may have been set previously with keyword \texttt{WCONPROD} or \texttt{WELTARG}. The well is still able to switch to other modes of control if necessary, and group control may also be applied in the usual way. The facility can be applied to just a subset of the production wells, if desired. Any wells using this facility must previously have been declared as producers with keyword \texttt{WCONPROD}. If the wells are placed under oil rate control in \texttt{WCONPROD}, an oil rate target/limit must still be specified in that keyword, but it is overwritten at the start of the next timestep.

The GOR penalty option can subsequently be turned off for individual wells by giving them a zero or negative Base GOR. Unless a new oil rate limit is specified with \texttt{WCONPROD} or \texttt{WELTARG}, it remains at the value last calculated. If no producers are using the facility, the timesteps are no longer restricted to terminate at the end of each calendar month.

The \texttt{WGORPEN} keyword should normally be applied at the beginning of a month. If it is entered part way through a month, the average GOR for that month (which is used to calculate the next month’s oil rate) reflects the production between the time of the keyword entry and the end of the month.

The keyword is followed by any number of records, each containing some or all of the following items of data and terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the \textit{ECLIPSE Technical Description} for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword \texttt{WLIST}.

2. Base GOR
   A zero or negative value turns the GOR Penalty option off for the specified well(s).
   \textbf{UNITS:} \(\text{sm}^3/\text{sm}^3\) (\textsc{metric}), Mscf/stb (\textsc{field}), scc/scc (\textsc{lab}), \(\text{sm}^3/\text{sm}^3\) (PVT-M)

3. Maximum Rate Limit
   \textbf{UNITS:} \(\text{sm}^3/\text{day}\) (\textsc{metric}), stb/day (\textsc{field}), scc/hr (\textsc{lab}), \(\text{sm}^3/\text{day}\) (PVT-M)
4. Initial average GOR

UNITS: \( \text{sm}^3/\text{sm}^3 \, (\text{METRIC}) \), \( \text{Mscf/stb (FIELD)} \), \( \text{scc/scc (LAB)} \), \( \text{sm}^3/\text{sm}^3 \, (\text{PVT-M}) \)

The value entered here is used to calculate the first month’s oil rate target.

This item may be defaulted. Its default value is set as follows:

- If the GOR penalty option is already in use, the previous month’s average GOR for each well will already have been calculated. It is left unchanged.

- If the GOR penalty option has just been turned on, the average GOR has not been calculated. It is therefore set equal to the well’s GOR at the previous timestep. If the well has just been opened, the average GOR defaults to the Base GOR set in item 2.

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

**Example**

```
WGORPEN
-- well name   base GOR   MRL   Initial average GOR
   PROD1      2.0        1000   /
   PROD2      1.8        1000   /
/
```
WGRUPCON  Sets well guide rates for group control

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Is the well available for group control?
   
   YES The well is automatically placed under group control if a rate target is imposed on its group, or on a higher level group to which it connects, or on the field.

   NO The well is prevented from coming under group control - it flows according to its own target or limits.

   DEFAULT: YES

3. Guide rate
   
   (A dimensionless number governing the well’s share of the group’s target rate)

   The target rates of the wells under group control are set in proportion to their guide rates, to meet the group or field production target. If the guide rate phase (see item 4) differs from the phase under control, the guide rate is translated into a guide rate for the controlled phase using the well’s flow rates at the beginning of each timestep.

   If a zero or negative guide rate is specified, the well’s guide rate is set at the beginning of each timestep from the general formula specified with keyword GUIDERAT. If a general formula has not been specified, or for injection wells (for which there is no general formula), the well’s guide rate is set equal to its production or injection potential (that is the flow rate the well would instantaneously achieve in the absence of any rate constraints; see "Well potentials" in the ECLIPSE Technical Description). When the well’s guide rate is set to its potential, the change in guide rate between timesteps is subject to limits defined in the keyword DGRDT for ECLIPSE 300.

   DEFAULT: Negative

4. Phase to which the well’s guide rate applies:
   
   OIL Guide rate applies to water phase

   WAT Guide rate applies to oil phase

   GAS Guide rate applies to gas phase

   LIQ Guide rate applies to liquid phase (for production wells only)

   COMB Guide rate applies to linearly combined phase (coefficients must be set with LINCOM) (for production wells only)

   CVAL Guide rate applies to the calorific production rate (for production wells only)

ECLIPSE 300 only
RAT  Guide rate applies to the surface flow rate of the phase that the well is injecting (for injectors only)

RES  Guide rate applies to the reservoir fluid volume rate

This item can be defaulted if the well’s guide rate has been defaulted or given a negative value.

DEFAULT: ‘ ’ = undefined

5. Guide rate scaling factor

   The well’s guide rate (either set in item 3 or calculated from its potentials) is multiplied by the number specified here.

   DEFAULT: 1.0

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

---

Note: This keyword is required only if well guide rates have to be set specifically or require a scaling factor, or if certain wells are to be prevented from coming under group control. If the keyword is not used, all wells are available for group control if required, and their guide rates are set from the formula specified with keyword GUIDERAT (if present), or set equal to their potentials (if keyword GUIDERAT is not present).

---

Example

```
WGRUPCON
 'P*' YES 100 LIQ /
 PROD2 NO /
```

---

Keywords
WH2NUM Two-phase WAG hysteresis saturation function region numbers

The WH2NUM saturation function region number specifies which set of water saturation functions (input using SWOF, SWFN and related keywords in the PROPS section) should be used for the two-phase water relative permeabilities associated with the WAG hysteresis option specified via the WAGHYSTR keyword.

The keyword should be followed by one integer for every grid block in the current input box specifying the saturation function region to which it belongs. The region number should not be less than 1 or greater than NTSFUN (see keyword TABDIMS). The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

This keyword is optional if the WAGHYSTR keyword has been specified. If omitted, the region numbers will default to those assigned to the SATNUM keyword. If specified, for grid blocks where the region number specified by WH2NUM is different to that specified by SATNUM, the WAG hysteresis two-phase water relative permeability functions will be taken from the WH2NUM table for saturation regions for which WAG hysteresis has been applied to the water phase.

For more information on the saturation keywords, see the SWFN and SWOF keywords in the PROPS section. The WAG hysteresis models are described in more detail under "Hysteresis in WAG floods" in the ECLIPSE Technical Description

Example

With NTSFUN=2 specified in keyword TABDIMS; NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS; and no input BOX set:

```
WH2NUM
144*1 96*2 /
```
## WH3NUM

### Three-phase WAG hysteresis saturation function region numbers

The WH3NUM saturation function region number specifies which set of water saturation functions (input using SWOF, SWFN and related keywords in the PROPS section) should be used for the three-phase water relative permeabilities associated with the WAG hysteresis option specified via the WAGHYSTR keyword.

The keyword should be followed by one integer for every grid block in the current input box specifying the saturation function region to which it belongs. The region number should not be less than 1 or greater than NTSFUN (see keyword TABDIMS). The data must be terminated by a slash (/).

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example 115*2). Note that spaces must not be inserted on either side of the asterisk.

This keyword is optional if the WAGHYSTR keyword has been specified. If omitted, the region numbers will default to those assigned to the IMBNUM keyword. If specified, for grid blocks where the region number specified by WH3NUM is different to that specified by IMBNUM, the WAG hysteresis three-phase water relative permeability functions will be taken from the WH3NUM table for saturation regions for which WAG hysteresis has been applied to the water phase.

For more information on the saturation keywords, see the SWFN and SWOF keywords in the PROPS section. The WAG hysteresis models are described in more detail under "Hysteresis in WAG floods" in the ECLIPSE Technical Description.

### Example

With NTSFUN=2 specified in keyword TABDIMS; NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword DIMENS; and no input BOX set:

```
WH3NUM
144*1 96*2 /
```
WHEDREFD  Sets well reference depth for calculation of hydraulic heads

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Reference depth.

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

   DEFAULT: cannot be defaulted

   End the data record with a slash (/).

   End the set of records with a blank record, containing just a slash.

   The reference depths for wells not specified with this keyword are defaulted to the depth specified by the HYDRHEAD keyword.

Example

```
WHEDREFD
  'p*'  250 /
  PROD2 300 /
/
```

Keywords

2590
Control mode instructions for history matching wells

This keyword contains instructions that influence the control of all history matching wells (see keywords WCONHIST and WCONINJH).

The first item enables you to override the history matching production well control modes, set in all subsequent WCONHIST keywords, with a single specified control mode. In the initial stages of history matching, the production wells may be on RESV control so that they produce the correct amount of total reservoir voidage before the water cut and gas oil ratio have been matched. But when these ratios have been matched, you may wish to rerun the simulation with the wells producing their observed oil rates. Instead of having to work through all the WCONHIST keywords changing RESV to ORAT, you may just enter the WHISTCTL keyword once before the first WCONHIST keyword.

The second item can be used to instruct ECLIPSE to stop the run if any history matching well has changed to BHP control at the end of the report step. By default, the BHP lower limit of each history matching production well is automatically set to atmospheric pressure when it is first declared in WCONHIST. Similarly, the BHP upper limit for history matching injection wells is initialized to a very high value when they are first declared in WCONINJH. Thus if a well violates this limit and changes to BHP control, it is a sign that the Kh or skin factor is very badly matched. You can set the BHP limit to a more reasonable value to make the test more sensitive, using the keyword WELTARG placed after the first WCONHIST (or WCONINJH) keyword in which the well appears.

The WHISTCTL keyword is followed by a single record containing the following items, terminated with a slash (/).

1. The overriding production well control mode to be applied to all wells specified in subsequent WCONHIST keywords:

   - ORAT  Wells are controlled by the observed oil rate
   - WRAT  Wells are controlled by the observed water rate
   - GRAT  Wells are controlled by the observed gas rate
   - LRAT  Wells are controlled by the observed liquid rate (oil + water)
   - CRAT  Wells are controlled by the observed linearly combined rate (coefficients must be set with LINCOM)
   - NGL   Wells are controlled by the observed NGL rate
   - RESV  Wells are controlled by the reservoir fluid volume rate calculated from the observed phase flow rates
   - BHP   Wells are controlled by bottom hole pressure
   - NONE  No control mode override is applied; the control modes specified in subsequent WCONHIST keywords will be honored

DEFAULT: NONE, giving no control mode override
Note: This keyword will only change the control mode of a well when the well is named in a subsequent WCONHIST keyword. It has no effect on previous WCONHIST keywords.

2. Flag to terminate the run if any history matching well has changed to BHP control:
   
   YES Terminate the run
   
   NO Continue the simulation with the well on BHP control
   
   DEFAULT: NO

Note: The run only terminates if a well switches to BHP control; if the user sets one or more wells to BHP control with WHISTCTL or WCONHIST the simulation will continue.

Examples

Example 1

To change the control mode of all history matching production wells to produce the observed amount of oil:

```
WHISTCTL
   ORAT /
```

Example 2

To terminate the run if a history matching well changes to BHP control (applying no control mode override):

```
WHISTCTL
   1* YES /
```
WHTEMP Specifications for tubing head temperature (THT) calculations

This keyword is used to specify the VFP table reference number for tubing head temperature calculations or to set the constant tubing head temperature value. This option is supported only for production wells and not for injection wells. Tubing head temperature values for a well may be examined using the summary keyword WHTT.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

   The well(s) must previously have been declared as a producer.

2. Production well VFP table number

   This number should refer to a production VFP table that contains Tubing Head Temperatures in the body of the table (see keywordVFPPROD).

   Each production VFP table must have a unique table number irrespective of whether the body of the table holds pressures or temperatures.

   A well must have both a pressure VFP table and a temperature VFP table specified for a Tubing Head Temperature to be calculated from a VFP table.

3. The constant tubing head temperature

   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

   Either item 2 or item 3 of this keyword should be specified, but not both.

End each data record with a slash (/).

End the set of records with a slash (/).

Example

```
WHTEMP
PA1 1* 40.0 /
PA2 1 /
PA3 2 1*/
/
```

Keywords

2593
Enter overall water composition

The data consists of a single composition for each reservoir equation of state region. See "Water components" in the ECLIPSE Technical Description for further details of the water component option.

Example

For two water components

```
WI
1.0 0.0
/
```
WINJEDET  Make-up of a lumped component in terms of its detailed components for an injection well

This keyword specifies the make-up of the fluid to be injected at a well in terms of the detailed components in a lumped simulation. Lumped simulations are enabled with the LUMPDIMS keyword in the RUNSPEC section; further details can be found in "Delumping and lumping" in the ECLIPSE Technical Description.

CAUTION: The order of the entries in the tables is very important and should match exactly the order in which the lumpings and detailed components were entered in the LUMPING keyword.

The data comprises a well name, followed by a fraction for each detailed component for each lumping. The fraction represents the proportion of the mole fraction of the carrier component to be associated with that tracer/detailed component. For each lumped phase the fractions must add up to 1.

Each entry must be terminated with a slash (/).

1. Well name
2. The fraction of the first lumped component to be associated with the first detailed component of this lumping.
3. The fraction of the first lumped component to be associated with the second detailed component of this lumping.
...
...
...

n The fraction of the second lumped component to be associated with the first detailed component of this lumping
...
...

item 2 and each subsequent item can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS. Care should be taken to ensure that the detailed components for each lumped component will sum to one. If they do not sum to one they will be normalized, which may give unintended results.

The keyword must be terminated with a slash (/)

Example

For the example in the LUMPING keyword:

```
WINJEDET
IN18  0.5  0.5
     0.25  0.25  0.5 /
INEAST  0.0  1.0
         0.0  0.5  0.5 /
/    
```
Specify the nature of injection gas

This keyword specifies the nature of the gas to be injected by a gas injection well. This should be used if WCONINJE is used instead of WELLINJE to specify control data for gas injection wells in compositional models. However, it is not necessary to use WINJGAS for wells subordinate to a group having gas injection controls for which the nature of its injected gas is defined (keyword GINJGAS); this will ensure that the injection stream will be that defined for the group. If a gas injection well under group control has its own fluid set, then this fluid, and not that defined for the group, will be injected at a rate controlled by the group.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Well name, well name template, well list or well list template.
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. A character string specifying the nature of the fluid to be injected.
   GAS  The composition of the injected gas is set to that of the field separator gas. However, it is generally better to specify the injected gas composition more precisely.
   From 2007.1, GAS is internally converted to GV (for the field separator) which has better reinjection and gas accounting behavior. OPTIONS3 item 113 can be used to recover the pre-2007.1 behavior.
   STREAM  The molar composition of the injected fluid is defined using the WELLSTRE keyword. The name of the stream must be entered in item 3.
   MIX  The molar composition of the injected fluid is defined as a mixture using either keyword WINJMIX or WINJORD. The name of the mixture must be entered in item 3.
   GV  The injected fluid is taken from the vapor production of a nominated group. The name of the group must be entered in item 3.
   WV  The injected fluid is taken from the vapor production of a nominated well. The name of the well must be entered in item 3.
   GRUP  The injected fluid is that defined for a superior group.

   DEFAULT: GRUP
   Only the first two characters are significant.

3. A character string is required if the composition of the injection fluid was specified using STREAM, MIX, GV or WV in item 2.
   In these cases this item is a character string specifying the name of the well stream, mixture, group or well that defines or supplies the injection stream.

Keywords
The available gas supply from a source group (or well) may be augmented by advance gas defined in keyword GADVANCE (or WADVANCE). Gas used for fuel (GRUPFUEL) and sales (GRUPSALE) is by default subtracted from a source group’s available gas supply for reinjection, but keyword WTAKEGAS may be used to alter the priorities for reinjection, fuel and sales.

4. The mnemonic for a well stream to be used as make-up gas.

This is only required if make-up gas is to be used to make up the injection target of the well named in item 1, if insufficient injection gas is available from the source defined in items 2 and 3.

If make-up gas is used, the injection calculation may have to be iterated to match the injection gas composition with the injection rate (see keyword GCONTOL).

5. The stage of the separator that defines the fluid composition for injection

The vapor from any stage of the separator may act as a source of injection fluid. The default value of 0 results in the use of the vapor from the whole separator as the injection fluid.

Note that the choice of separator stage also affects the available produced gas supply rate from the source group or well (item 3) if the source is GV or WV (item 2).

DEFAULT: 0

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also keyword WAVAILIM.

Notes

- For accurate group gas accounting, there is a limitation that reinjection control and any sales gas or fuel gas requirements must all be specified for the same group, usually the FIELD group, and not split across the group hierarchy. This limitation only applies if the injection gas contains group vapor, with the vapor taken from a separator stage (that is not from the whole separator).

- When the fluid composition is taken from the produced vapor from another well or group of wells (item 2 set to WV or GV), then it is possible that the composition will be undefined, for example when the producing well or group is shut. In this case, for compositional fluids, the injection composition will be set to be 100% component 1.

Example

All injection wells are to use the vapor produced by the FIELD group.

```
WINJGAS
'\'I\' GV Field /
```

Keywords
WINJMIX specifies an injection mixture by fraction

WINJMIX specifies a number of fluid sources to be used to define an injection stream. The contribution of each fluid to the mixed injection stream is defined by a mixing fraction.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/). Each record defines a single fluid in a mixture.

The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form $n^*$, where $n$ is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. The name of the mixture.
   The maximum number of mixtures is set in WELLDIMS item 7 and is defaulted to 5.

2. The mixture contribution index of the fluid defined below.
   The maximum number of mixture fluids is set in WELLDIMS item 9 and is defaulted to 3. The contribution index must be an integer between 1 and this number.
   Note that the calculation of the injection mixture is not dependent on the order in which the mixture sources are specified.

3. The fraction of this fluid in the mixture.
   This fraction multiplies the molar composition of this fluid in the mixture.

4. The nature of this fluid.
   The fluid type must be one of the following:

   GAS    The composition of the fluid is set to that of the field separator gas. However, it is generally better to specify the injected gas composition more precisely.
   From 2007.1, GAS is internally converted to GV (for the field separator) which has better reinjection and gas accounting behavior. item 113 in OPTIONS3 can be used to recover the pre-2007.1 behavior.

   STREAM The molar composition of the fluid is defined using the WELLSTRE keyword. The name of the stream must be entered in item 5.

   GV     The fluid is taken from the vapor production of a nominated group. The name of the group must be entered in item 5.

   WV     The fluid is taken from the vapor production of a nominated well. The name of the well must be entered in item 5.

   Only the first two characters are significant.

5. A character string is required if the fluid was specified as STREAM, WV or GV.
   In these cases this item specifies the name of the wellstream, well or group that defines the fluid’s composition.

6. The stage of the separator that defines the fluid composition.
The vapor from any stage of the separator may act as a source of injection fluid. The default value of 0 results in the use of the vapor from the whole separator as the injection fluid.

Note that the choice of separator stage also affects the available produced gas supply rate from the source group or well (item 5) if the source is GV or WV (item 4).

DEFAULT: 0

Notes

- If the WINJMIX keyword contains only well or group vapor sources and the WAVAILIM keyword is present, the injection rate of the mixture is limited to the sum of the well and group vapor production. However, availability does not extend to items within the mixture; if a mixture fraction is set that requires a well or group to provide more vapor than it produces, then that well or group over-injects and a negative excess will be set in the gas accounting tables. Essentially, WINJMIX provides a mixture whose composition is independent of the injection rate. If you wish to honor the availability of individual items within the mixture, WINJORD should be used instead.

- Prior to 2008.1 if a mixture contained a well or group vapor item and this well or group had advanced gas associated with it, then the advanced gas was combined with the available produced gas of that well or group. The advanced gas and the available produced gas were treated as one item in the mixture; note that the advanced gas was not used before the produced gas in this case, regardless of the setting of item 30 in OPTIONS3 item 30. From 2008.1 this behavior was altered so that advanced gas is injected before the mixture gas in the same manner as other reinjection sources. The pre-2008.1 behavior may be recovered using item 139 of OPTIONS3.

- If a stream is present as an item in the mixture, this effectively has an infinite supply and thus the availability of the mixture is infinite. The well injection fraction table (if present) will show values that do not sum to unity; the difference is accounted for by the stream item in the mixture. Note that the composition of the mixture is independent of the injection rate, and that the availability of individual GV or WV items also present in the mixture are not honored, which could therefore over inject and show a negative excess in the gas accounting tables. This situation is handled better using WINJORD instead.

- For accurate group gas accounting, there is a limitation that reinjection control and any sales gas or fuel gas requirements must all be specified for the same group, usually the FIELD group, and not split across the group hierarchy. This limitation only applies if the injection mixture contains a group vapor item, with the vapor taken from a separator stage (that is not from the whole separator).

Example

```
WINJMIX
  MIXA 1 0.4 WV P1 /
  MIXA 2 0.4 WV P2 /
  MIXA 3 0.2 ST ST1 /
```

Keywords
WINJMULT Pressure-dependent injectivity multipliers

This keyword may be used to provide multipliers to the injectivity index of injection wells that vary with the wells’ BHP or the pressure in the wellbore adjacent to the individual connections. The injectivity multipliers could be used to take approximate account of hydraulic fracturing at water injection wells.

Each time the well flows are calculated, the fluid mobility at each connection in the well is multiplied by a factor:

\[ 1.0 + a \left( P_w - P_f \right) \quad \text{when} \quad P_w > P_f \quad \text{or} \]

\[ 1.0 \quad \text{when} \quad P_w < P_f \]

where

\[ a \] is the multiplier gradient set in item 3

\[ P_w \] is the instantaneous value of the well’s bottom hole pressure or the wellbore pressure adjacent to the individual connections (see item 4)

\[ P_f \] is the fracturing pressure set in item 2.

The injectivity multipliers may be either reversible or irreversible, depending on the setting of item 4. If the irreversible option is selected, the multipliers do not decrease if the well pressure subsequently decreases.

The multipliers act on the fluid mobilities and thus increase the injectivity index of the well. The reported values of the connection transmissibility factor remain unaltered.

The multipliers are only applied to injection wells. If any production wells are specified here, their productivity index is not altered.

Wells with pressure-dependent injectivity multipliers cannot also have a D-factor (flow-dependent skin). Also, do not set their injectivity index using keyword WELPI, because this calculation is based on the original fluid mobilities at each connection in the well. The well’s PI would only be correctly set to the user supplied value if fracturing had yet to occur.

This keyword has not been implemented within the Multisegment Well Model.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Fracturing pressure (\( P_f \))

   UNITs: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

   This can be defaulted only when item 3 is defaulted.

3. Multiplier gradient (\( a \))
UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT–M)
DEFAULT: 0.0

4. Mode of operation

\textbf{WREV} All connections in the well have the same injectivity multiplier, which is determined by the well’s BHP. items 5 - 7 are ignored. The increase in injectivity is reversed if the BHP subsequently declines.

\textbf{CREV} The injectivity multiplier applies to the set of connections defined by items 5 - 7, and is determined by the wellbore pressure adjacent to each connection. The increase in injectivity is reversed for \textbf{all} connections in the well if the wellbore pressure subsequently declines.

\textbf{CIRR} The injectivity multiplier applies to the set of connections defined by items 5 - 7, and is determined by the wellbore pressure adjacent to each connection. The increase in injectivity is \textbf{irreversible} for \textbf{all} connections in the well; each connection continues using its maximum attained injectivity multiplier if the wellbore pressure declines.

DEFAULT: WREV

5. I - location of connecting grid block(s)
DEFAULT: Negative (allows any I-location)

6. J - location of connecting grid block(s)
DEFAULT: Negative (allows any J-location)

7. K - location of connecting grid block(s)
DEFAULT: Negative (allows any K-location)

If items 5 - 7 are all defaulted, or item 4 set to \textbf{WREV}, the injectivity multiplier will be applied to all connections in the well.

If any of items 5 - 7 are set positive, and item 4 is \textbf{not} set to \textbf{WREV}, the injectivity multiplier applies only to a subset of the connections in the well. The connections in the subset are those with I,J,K locations matching the specified location indices in items 5 - 7. A zero or negative value for a location index (I, J or K) matches any value of that location index.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

\textbf{Example}

Injection wells IW1 and IW2 use the default \textbf{WREV} mode, in that all their connections have the same injectivity multiplier, which depends on the well’s BHP and is reversible if the BHP declines. For injection well IW3, the connections in layers 3 and 4 have their own separate injectivity multiplier parameters and the increase in injectivity index in this well is irreversible.

\begin{verbatim}
WINJMULT
IW1 5000 0.0052 /
IW2 4800 0.013 /
IW3 4600 0.0045 CIRR 0 0 3 /
IW3 4700 0.0042 CIRR 0 0 4 /
\end{verbatim}
**WINJOIL**

**Specify the nature of injection oil**

This keyword specifies the nature of the oil to be injected by an oil injection well and should be used in conjunction with the `WCONINJE` keyword (for compositional and thermal models).

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Well name, well name template, well list or well list template.
   - A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the *ECLIPSE Technical Description* for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword `WLIST`.

2. A character string specifying the nature of the fluid to be injected.
   - `STREAM` The molar composition of the injected fluid is defined using the `WELLSTRE` keyword. The name of the stream must be entered in item 3.
   - DEFAULT: `STREAM`
     - Only the first two characters are significant.

3. A character string is required specifying the name of the well stream that defines the injection stream.
   - End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

**Example**

For a two-component model:

```
WELLSTRE 'OIL1' 0.5 0.5 /
WINJOIL 'I*' STREAM OIL1 /
```

---

**Keywords**
WINJORD specifies a number of fluid sources to be used to define an injection stream. It specifies an ordering of the fluids. These are taken in the specified order subject to their availability.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/). Each record defines a single fluid in the mixture.

The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. The name of the mixture.
   The maximum number of mixtures is set in WELLDIMS item 7 and is defaulted to 5.

2. The mixture contribution index of the fluid defined below.
   The maximum number of mixture fluids is set in WELLDIMS item 9 and is defaulted to 3. The contribution index must be an integer between 1 and this number.
   The fluids will be taken in increasing order of their contribution index, from 1 to n.

3. The nature of this fluid.
   The fluid type must be one of the following:
   
   GAS The composition of the fluid is set to that of the field separator gas. However, it is generally better to specify the injected gas composition more precisely.
   
   From 2007.1, GAS is internally converted to GV (for the field separator) which has better reinjection and gas accounting behavior. item 113 of OPTIONS3 can be used to recover the pre-2007.1 behavior.

   STREAM The molar composition of the fluid has been defined using the WELLSTRE keyword. The name of the stream must be entered in item 4.

   GV The fluid is to be taken from the vapor production of a nominated group. The name of the group must be entered in item 4.

   WV The fluid is to be taken from the vapor production of a nominated well. The name of the well must be entered in item 4.

   Only the first two characters are significant.

4. A character string is required if the fluid was specified as STREAM, WV or GV.
   In these cases this item specifies the name of the wellstream, well or group that defines the fluid’s composition.

5. The stage of the separator that defines the fluid composition.
   The vapor from any stage of the separator may act as a source of injection fluid. The default value of 0 results in the use of the vapor from the whole separator as the injection fluid.

   Note that the choice of separator stage also affects the available produced gas supply rate from the source group or well (item 4) if the source is GV or WV (item 3).
Notes

- If a mixture contains a well or group vapor item and this well or group has advanced gas associated with it, then the advanced gas is combined with the available produced gas of that well or group. The advanced gas and the available produced gas are treated as one item in the mixture. Note that the advanced gas is not used before the produced gas in this case, regardless of the setting item 30 in OPTIONS3 keyword.

- If a stream is present as an item in the mixture, this effectively has an infinite supply and thus the availability of the mixture is infinite. The well injection fraction table (if present) shows values that do not sum to unity; the difference is accounted for by the stream item in the mixture. Note that, because of the infinite availability of a stream, no item entered after a stream is used.

- The composition of the injected gas in general varies with the injection rate, and thus the injection calculation may have to be iterated to match the injection gas composition with the injection rate (see keyword GCONTOL).

- For accurate group gas accounting, there is a limitation that reinjection control and any sales gas or fuel gas requirements must all be specified for the same group, usually the FIELD group, and not split across the group hierarchy. This limitation only applies if the injection mixture contains a group vapor item, with the vapor taken from a separator stage (that is not from the whole separator).

Example

```
WINJORD
MIXA 1 WV P1 /
MIXA 2 WV P2 /
MIXA 3 ST ST1 /
```

Keywords

2604
WINJTEMP

The nature of injection fluid

This keyword can only be used with the THERMAL option or the TEMP option.

WINJTEMP specifies the nature of the water, gas or steam gas to be injected by a well. The well control must have already been defined using the WCONINJE keyword. The pair of keywords WCONINJE and WINJTEMP are an alternative to using the WELLINJE keyword.

In thermal simulations, where an energy equation is solved, the specific enthalpy of the injected fluid must be specified. This can be done directly, by specifying the enthalpy at a given pressure (using items 4 and 5). This is not straightforward since the fluid enthalpy is not easy for the user to calculate. Alternative data can therefore be specified, allowing the simulator to calculate the enthalpy. This can be done by specifying the temperature at a given pressure (using items 3 and 4). However, if a mixture of water and steam is being injected (that is the pressure is the water saturation pressure) then the pressure and temperature do not specify the enthalpy uniquely. This is because different proportions of steam and water can exist at the saturation pressure and temperature. In this case the steam quality should be specified, as well as either the temperature (items 2 and 3) or the pressure (items 2 and 4). For further information regarding the calculation of the enthalpy of the injected fluid see "Water, steam and gas injection" and "Energies and enthalpies" in the ECLIPSE Technical Description.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated by a slash (/).

The records can be terminated early using the slash (/) character, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Well name, well name template, well list or well list template.

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

   This item is always required and the wells referenced must have already been defined by this point in the SCHEDULE section.

2. The steam quality of the injected fluid. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

   If specified, the steam quality must be between 0 and 1. Also, either the temperature or the pressure must be specified, but not the specific enthalpy rate.

   For water injectors or 100% gas injectors, the steam quality should not be used. For these injectors, specify the pressure and temperature (or pressure and enthalpy) instead.

3. The temperature of the injected fluid. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

   This quantity is used to define the specific enthalpy of the injected fluid. The actual temperature of the fluid at a particular point along the wellbore will depend on the pressure at that point. If the pressure in the wellbore changes over time, the temperature will therefore also change over time. See "Temperature at the bottom hole pressure" in the ECLIPSE Technical Description for further information.
If specified, either the steam quality or the pressure must also be specified, but not the specific enthalpy rate.

UNITs: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

4. The pressure of the injected fluid. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

If specified, then one of steam quality, temperature, or specific enthalpy rate must also be specified.

UNITs: barsa (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)

5. The specific enthalpy of the injected fluid. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

If specified, the pressure must also be specified. If specified, neither the steam quality nor the temperature may be specified.

UNITs: kJ/kg-M (METRIC), Btu/lb-M (FIELD), J/gm-M (LAB), kJ/kg-M (PVT-M)

Note: For 2012.1, the enthalpy calculation was improved. To revert this new calculation to pre-2012.1 behavior, set item 276 in OPTIONS3 to 1.

Example

```
WINJTEMP
  -- SQ T(C) P(BAR)
WELL1 0.7 200 1* / STEAM-WATER
WELL2 0.7 1* 15 / STEAM-WATER
WELL3 1* 20 15 / WATER
WELL4 1* 250 15 / STEAM
```
WINJW

Specify the nature of injection water

This keyword specifies the nature of the water to be injected by a water injection well, and is activated within the multi-component water model and options CO2STORE, GASWAT and GASSOL.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. Well name, well name template, well list or well list template
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. A character string specifying the nature of the fluid to be injected.
   
   STREAM The molar composition of the injected fluid is defined using the WELLSTRW keyword. The name of the stream must be entered in item 3.

   DEFAULT: STREAM
   
   Only the first two characters are significant.

3. A character string is required specifying the name of the well stream that defines the injection stream.
   
   End data record with a slash (/).

   End the set of records with a blank record, containing just a slash.

See "Water components” in the ECLIPSE Technical Description for further details of the water component option.

Example

```
WINJW
'1*' STREAM Brine/
```
WINJWAT  The nature of injection water

This keyword can only be used with the THERMAL option (keyword THERMAL in the RUNSPEC section).

WINJWAT is an alias for WINJTEMP.
Automatic retubing and THP or lift switching data

The keyword supplies data for the automatic retubing and THP or lift switching option. If a well's production rate falls below the value set in item 2, or if its water cut exceeds the value set in item 7, or if its gas-liquid ratio exceeds the value set in item 9, or if the well dies under THP control, then one of three operations is performed:

- If a THP value has been set in item 8, the well switches to this new THP limit in decrements defined by item 11. This can be regarded as switching the well into a lower pressure separator to increase its productivity. Obviously, to have the desired effect, the new THP limit must be lower than the well's original THP limit. If a decrement is set in item 11 the THP switching is performed in stages; each time THP switching is triggered the THP is reduced by the specified decrement, until the final THP value set in item 8 is reached.

- If a VFP table number has been set in item 4, the well switches to this table. This can be regarded as a retubing operation, if the new table has been calculated with a different tubing diameter from the original table.

- If an artificial lift quantity has been set in item 5, the well uses this new lift quantity to interpolate the VFP table. The new ALQ will be reached in increments defined in item 10. This can be regarded as switching on (or uprating) artificial lift in the well. If an increment is set in item 10 the lift switching is performed in stages; each time lift switching is triggered the ALQ will be increased by the specified increment, until the final ALQ value set in item 5 is reached. If an efficiency factor has been set in item 6, the well uses this new value in place of its original value when the lift switching (or its first increment) has taken place, to allow for an increase in downtime resulting from the lift system. The switching of artificial lift can be made subject to constraints on the number of wells using lift, and the sum of their ALQs, by using the keyword GLIFTLIM. (Alternatively, the new ALQ value in item 5 could be lower than the well’s original value, as long as it is still above 0.0, if artificial lift is to be reduced when the well’s formation GLR increases beyond the limit set in item 9.)

- If positive values are specified for two or more of items 8, 4 and 5, a multistage process will result. A switching operation is performed each time the well’s production rate, water cut or GLR violates a limit set in item 2, 7 or 9. The operations are performed in the following order:
  - Switch to the lower THP limit (for example change to a lower pressure separator)
  - Switch to the second VFP table (for example perform a retubing operation)
  - Switch to the new artificial lift quantity (for example turn on lift).

**Note:** The order of the above actions can be overridden if the PRORDER keyword is used. By setting item 2 in record 2 of keyword PRORDER to 'YES', the order given above is replaced by the order given in the first record of PRORDER.

The switching operations are performed at the end of the timestep in which the minimum rate or maximum water cut or GLR limit is violated. But if a limit tolerance fraction has been set with keyword WLIMTOL, the timestep is recalculated if the limit is violated by more than this fraction.

The WLIFT keyword may also be used to enter secondary values of the THP, the VFP table number and the ALQ to be activated by the group production rules (see actions THP, RETUBE and LIFT in the PRORDER keyword, and "Group production rules" in the ECLIPSE Technical Description). Switching is performed when required to keep the well’s group (or a superior group) producing at its target rate set in keyword GCONPROD. THP or lift switching is performed in stages if increments are set in items 11 or 10. The order in which the different switching operations occur in these circumstances depends upon the order
in which the actions are listed in the PRORDER keyword. If you wish the switching process to be activated only by the Group Production Rules or if the well dies under THP control, and not by the rate, water cut or GLR of the well itself, then you must default the rate, water cut and GLR limits in items 2, 7 and 9 of this keyword.

The WLIFT keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated at any item after item 1. The remaining items assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Well production rate below which the switching operations are triggered
   A zero or negative value implies that the switching operations are not triggered by a low well production rate. Default this item if you require the switching operations to be triggered only by the group production rules (keyword PRORDER) or if the well dies under THP control.
   UNITS: sm$^3$/day (METRIC), stb/day for OIL or LIQUID, or Mscf/day for GAS (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)
   DEFAULT: 0.0

3. Phase to which the production rate limit in item 2 refers
   OIL, LIQUID, or GAS
   (The word may be abbreviated; only the first character is significant)
   DEFAULT: OIL

4. New VFP table number for retubing
   When retubing is performed this will replace the table number previously specified with keyword WCONPROD.
   DEFAULT: 0
   No retubing operation is performed if a default, zero or negative value is entered.

5. New artificial lift quantity for lift switching
   When lift switching is performed this replaces the artificial lift quantity previously specified with keyword WCONPROD or WELTARG. The lift switching is be performed in stages if an incremental value is set in item 10.
   UNITS: These depend on what the ALQ was meant to represent when the table was created. Refer to section "Record 6:ALQ values" in the description of keyword VFPPROD.
   DEFAULT: 0.0
   No lift switching operation is performed if a default, zero or negative value is entered.
6. New well efficiency factor after lift switching

When lift switching is performed this will replace the efficiency factor previously specified with keyword \texttt{WEFAC}, for example to account for any increase in downtime resulting from the lift system. If lift switching is performed in stages, the new efficiency factor takes effect when the first ALQ increment is applied.

DEFAULT: 0.0

A zero or negative value leaves the original efficiency factor unchanged.

7. Well water cut value above which the switching operations are triggered

A zero or negative value implies that the switching operations are not triggered by a high well water cut. Default this item if you require the switching operations to be triggered only by the group production rules (keyword \texttt{PRORDER}) or if the well dies under THP control.

UNITS: \texttt{sm}^3/\texttt{sm}^3 (\textsc{metric}), \texttt{stb/stb} (\textsc{field}), \texttt{scc/scc} (\textsc{lab}), \texttt{sm}^3/\texttt{sm}^3 (\textsc{pvt-m})

DEFAULT: 0.0

8. New THP limit

When THP switching is performed, this replaces the THP limit previously specified with keyword \texttt{WCONPROD} or \texttt{WELTARG}. The THP switching is performed in stages if an incremental value is set in item 11.

UNITS:\texttt{barsa} (\textsc{metric}), \texttt{psia} (\textsc{field}), \texttt{atma} (\textsc{lab}), \texttt{atma} (\textsc{pvt-m})

DEFAULT: 0.0

No THP switching operation is performed if a default or zero value is entered.

\textit{Note:} Switching to a lower THP limit has no effect if the well’s group is part of a network (see “Network option” in the \textsc{eclipse} Technical Description), as the wells’ THP limits are set by their group’s nodal pressure in the network.

9. Well gas-liquid ratio value above which the switching operations are triggered

A zero or negative value implies that the switching operations are not triggered by a high GLR. Default this item if you require the switching operations to be triggered only by the group production rules (keyword \texttt{PRORDER}) or if the well dies under THP control.

UNITS: \texttt{sm}^3/\texttt{sm}^3 (\textsc{metric}), \texttt{Mscf/stb} (\textsc{field}), \texttt{scc/scc} (\textsc{lab}), \texttt{sm}^3/\texttt{sm}^3 (\textsc{pvt-m})

DEFAULT: 0.0

10. Increment to be added to (or subtracted from) the well’s ALQ value at each lift switching event

Each time lift switching is triggered for the well, its ALQ is changed by the specified increment, until the final ALQ value set in item 5 is reached.

UNITS: These depend on what the ALQ was meant to represent when the table was created. Refer to section “Record 6: ALQ values” in the description of keyword \texttt{VFPPROD}.

DEFAULT: 1.0E20 (infinity, that is \textsc{eclipse} will switch to the secondary value in item 5 in a single operation.)

11. Decrement to be subtracted from the well’s THP value at each THP switching event (entered as a positive number)
Each time THP switching is triggered for the well, its THP is decreased by subtracting the specified decrement, until the final THP value set in item 8 is reached.

**UNITS:** bars (**METRIC**), psi (**FIELD**), atm (**LAB**), atm (**PVT-M**)

**DEFAULT:** 1.0E20 (infinity, that is ECLIPSE will switch to the secondary value in item 8 in a single operation.)

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Switching operations will not be performed on a well whose rate target is currently cut back under prioritization, to prevent these happening prematurely while the well is temporarily flowing with a small rate target.

See also keywords **GLIFTLIM**, **WLIMTOL** and **PRORDER**.

**Example**

Well **PROD1** is given an artificial lift quantity of 5.0 and an efficiency factor of 0.8 when its water cut first exceeds 0.5.

Well **PROD2** has its VFP table number changed to 3 the first time its liquid production rate falls below 500, and on the second time it is given artificial lift as **PROD1**.

Well **PROD3** has its THP limit reduced to 60, in steps of 20, whenever its liquid production rate falls below 500. When the THP limit has been reduced to 60, its VFP table number is changed to 4 the next time its liquid rate falls below 500.

<table>
<thead>
<tr>
<th>WLIFT</th>
<th>--well</th>
<th>min flow</th>
<th>new VFP</th>
<th>new eff fac</th>
<th>WCT</th>
<th>max</th>
<th>new ALQ</th>
<th>max THP</th>
<th>ALQ incr</th>
<th>ALQ decr</th>
</tr>
</thead>
<tbody>
<tr>
<td>PROD1</td>
<td>1*</td>
<td>1*</td>
<td>0</td>
<td>5.0</td>
<td>0.8</td>
<td>0.5</td>
<td>/</td>
<td>/</td>
<td>/</td>
<td>/</td>
</tr>
<tr>
<td>PROD2</td>
<td>500</td>
<td>LIQ</td>
<td>3</td>
<td>5.0</td>
<td>0.8</td>
<td></td>
<td>/</td>
<td>/</td>
<td>/</td>
<td>/</td>
</tr>
<tr>
<td>PROD3</td>
<td>500</td>
<td>LIQ</td>
<td>4</td>
<td>1*</td>
<td>1*</td>
<td>60</td>
<td>1*</td>
<td>1*</td>
<td>20</td>
<td>/</td>
</tr>
</tbody>
</table>

/
WLIFTOPT Gas lift optimization data for wells

The WLIFTOPT keyword must be used to tell ECLIPSE which wells should have their lift gas injection rates calculated by the optimization facility; and to set their maximum and minimum lift gas injection rates and weighting factors. The LIFTOPT keyword must be entered before WLIFTOPT, to activate the Gas Lift Optimization facility.

See "Gas lift optimization" in the ECLIPSE Technical Description for further details.

The keyword is followed by any number of records, each containing between two and seven items of data and terminated with a slash (/). The set of records must end with a blank record, containing only a slash.

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Is the well’s lift gas injection rate to be calculated by the optimization facility?
   YES The well’s lift gas injection rate is calculated by the optimization facility.
   NO The well’s lift gas injection rate remains fixed at a value that can be set either in item 3 of this keyword, or in item 12 of keyword WCONPROD, or with keyword WELTARG.

3. Maximum rate of lift gas injection for the well
   To prevent extrapolation of the VFP tables, any value entered here must not exceed the largest ALQ value in the well’s VFP table.
   If item 2 is NO, then item 3 is regarded as the fixed lift gas injection rate for the well. If item 3 is defaulted, the lift gas rate remains unchanged at its current value.
   UNITS: sm$^3$/day (METRIC), Mcf/day (FIELD), scf/hr (LAB), sm$^3$/day (PVT-M)
   DEFAULT: The largest ALQ value in the well’s VFP table (if item 2 is YES). Remain unchanged (if item to is NO).

4. Weighting factor for preferential allocation of lift gas $\alpha_w$
   Each increment of lift gas supply is allocated to the well that currently has the largest weighted incremental gradient which is calculated according to the formula $\alpha_w \cdot \Delta Q_o / (\Delta Q_{gl} + \alpha_g \cdot \Delta Q_g)$
   where $\alpha_w$ is the weighting factor for the preferential allocation of lift gas, $\Delta Q_o$ and $\Delta Q_g$ are the increments in oil and gas rate respectively for a gas lift increment of $\Delta Q_{gl}$ and $\alpha_g$ is the incremental gas rate weighting factor defined by item 6 below. Further lift gas increments are not allocated to the well if its weighted incremental gradient is less than the minimum economic gradient set in item 2 of keyword LIFTOPT.
   This item is ignored if item 2 is NO.
   DEFAULT: 1.0

5. Minimum rate of lift gas injection for the well
If a positive value is specified, the well is allocated at least that amount of lift gas, unless the well is unable to flow with that rate of lift gas injection, or unless the well can already meet one of its own rate limits before receiving its minimum lift gas rate.

If a negative value is specified, the well is allocated at least enough lift gas to enable it to flow, provided that it can flow at a lift gas injection rate within the maximum set in item 3. The magnitude of the negative value is not important.

If there is not enough lift gas available to satisfy all the wells’ minimum requirements, the wells are allocated their minimum requirements in decreasing order of their weighting factor (item 4).

If the well belongs to a group (including the FIELD) having a production rate target that can be achieved without any gas lift, by default no lift gas is allocated to the well, regardless of any minimum requirement. However, if the well is given a weighting factor greater than 1.0 (item 4), the well receives its minimum lift gas requirement even if the group could achieve its production rate target without any gas lift, unless the well can flow at one of its own rate limits without its full minimum amount of lift gas.

This item will be ignored if item 2 is NO.

UNITS: sm$^3$/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: 0.0

6. Incremental gas rate weighting factor $\alpha_g$

The incremental gradient is calculated according to the formula $\alpha_w \cdot \Delta Q_o / (\Delta Q_{gl} + \alpha_g \cdot \Delta Q_g)$

where $\alpha_w$ is the weighting factor for the preferential allocation of lift gas defined by item 4 above, $\Delta Q_o$ and $\Delta Q_g$ are the increments in oil and gas rate respectively for a gas lift increment of $\Delta Q_{gl}$ and $\alpha_g$ is the incremental gas rate weighting factor. If this value is greater than zero, the incremental gas rate will influence the calculation of the incremental gradient and may be used to discourage the allocation of lift gas to wells which produce more gas.

This item is ignored if item 2 is NO.

DEFAULT: 0.0

7. Allocate additional lift gas when a group gas target has been achieved but the oil rate limit has not been reached?

By default, unless an overriding minimum lift gas has been defined in item 5 above, during the gas lift optimization process a well which would potentially cause a group gas target to be exceeded, if allocated further lift gas, will be constrained from receiving more lift gas.

If this item is defined to be YES this constraint is removed for a group which is on gas target control and subject to an oil rate limit. This provides a mechanism for continuing gas lift optimization to increase the oil rate, up to the oil rate limit, after a group gas rate target has been achieved. Following the completion of the gas lift optimization calculations, however, any potential improvement in the oil rate of an individual well which has been achieved in this manner may be lost at the expense of additional lift gas if the well production is reduced subsequently to comply with the group constraints.

This item is ignored if item 2 is NO.

YES  The well can receive further lift gas.

NO  The well cannot receive further lift gas.
DEFAULT: NO

End the data record with a slash (/).

End the set of records with a blank record containing just a slash (/).

**Example**

<table>
<thead>
<tr>
<th>WLIFTOPT</th>
<th>-- well</th>
<th>optimize</th>
<th>max lift</th>
<th>weighting</th>
<th>min lift</th>
<th>gas lift?</th>
<th>gas rate</th>
<th>factor</th>
<th>gas rate</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-- name</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>SM3/DAY</td>
<td></td>
<td></td>
<td>SM3/DAY</td>
</tr>
<tr>
<td>'PA*'</td>
<td>YES</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>'PB*'</td>
<td>YES</td>
<td>1*</td>
<td>1.01</td>
<td>-1.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PC1</td>
<td>NO</td>
<td>0.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PC2</td>
<td>NO</td>
<td>5E4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Example:

|        |          |          |          |           |          |          |          |        |          |
| 'PA*'  | YES      |          |          |           |          |          |          |        |          |
| 'PB*'  | YES      | 1*       | 1.01     | -1.0      |          |          |          |        |          |
| PC1    | NO       | 0.0      |          |           |          |          |          |        |          |
| PC2    | NO       | 5E4      |          |           |          |          |          |        |          |

ECLIPSE Reference Manual

Keywords

2615
WLIMTOL

Tolerance fraction for economic and other limits

The keyword should be followed by a single real number that represents the tolerance fraction, terminated with a slash (/).

The tolerance fraction applies to the following limits:

- Connection, well, group and field economic limits (see keywords CECON, WECON, GECON and WECONINJ),
- Group/field maximum rate limits resulting in well workovers or closures (see keyword GCONPROD and GCONPRI),
- Well and group cutback limits (see keywords WCUTBACK and GCUTBACK).

ECLIPSE 100:

- Automatic retubing and lift switching limits (see keyword WLIFT),
- Maximum THP design limits (see keyword WTHPMAX),
- Bottom hole GLR limits (see keyword WBHGLR).

ECLIPSE 100

Note: In particular this keyword does not apply to the keywords CECONT, WECONT, GECONT, WCUTBACT and GCUTBACT which always repeat the timestep.

If one of these limits is violated during a timestep, the remedial action is taken at the end of the timestep. Thus the limit is broken for one timestep before the action is taken. However, if the limit is broken by more than the tolerance fraction multiplied by the limiting value, the timestep is recalculated after the remedial action has been taken. In this case, the action takes effect from the beginning of the timestep during which the limit would otherwise have been violated.

Note, however, that the timestep is repeated only once; if a limit is still violated at the end of the repeated timestep another remedial action is taken but the timestep is not repeated again.

If the keyword is not used, the tolerance fraction is set to infinity (1.0E20), and the timestep is not recalculated for any limit violation.

ECLIPSE 100

See also keyword GCONTOL, regarding the tolerance applied to group control targets.

Example

```
WLIMTOL
0.2 /
```
WLIST  

Sets up lists of well names, for use in other well keywords

The keyword WLIST may be used to construct and maintain well lists, which are lists of well names. Well lists whose names begin with an asterisk (*) may be entered wherever a well name root can be entered in the SCHEDULE section well keywords, as an alternative method of specifying several wells at once.

Each list should have a unique name, consisting of up to eight characters. The first character must be an asterisk and the second character must be a letter. Each list may contain any number of well names. Up to 50 well names may be added to a list in a single data record. Lists containing more than 50 well names may be constructed by appending well names to the list in subsequent data records. Wells may be added to, deleted from or moved between lists at any time in the simulation. A well may belong to more than one list at a time, up to a maximum as set in item 11 of WELLDIMS.

In ECLIPSE 100, well lists are also used by keyword WLISTARG to set individual operating targets or limits for several wells in a single data record. The WLIST keyword supersedes the old WLISTNAM keyword, which is retained for back compatibility. WLISTNAM must still be used to operate on old-style list names (beginning with a letter), but these lists are not recognized in any keyword other than WLISTARG.

Well lists may be used in the ACTIONX keyword to define a grouping of wells, whose combined rates, totals or ratios can form part of a triggering condition in this keyword.

Combined rates, totals and ratios for all the wells in a well list may also be written to the summary file using an appropriate group summary mnemonic followed by the well list name (see Table 2.28).

The WLIST keyword should be followed by any number of data records, terminating with a blank record containing just a slash (/). Each data record consists of the following items, and is terminated with a slash. The data items are:

1. The well list name.
   This should consist of up to 8 characters, enclosed in quotes. The first character of the list name must be an asterisk and the second character must be a letter.

2. The operation.
   This must be one of:

   NEW  Starts a fresh list having the specified name. If a list with that name already exists, all well names are first removed from it. The following well names are then placed on the list.

   ADD  Appends the following well names to the list.

   MOV  Moves the following wells from whatever list they are currently a member of to the list named in item 1, appending them to the end of this list.

   DEL  Removes the following well names from the list named in item 1.

3. A list of up to 50 well names.
   The wells must previously have been declared with the WELSPECS keyword.

Note: Well list totals are the cumulative total of wells currently on the list. If a well is removed from the list then its contribution the total will also be removed. Similarly, if a well is added to the list then the full well total will be included, not the total from the time at which the well was added.
Well name roots (enclosed in quotes and ending with an asterisk) may also be used here. When a well name root is encountered, all currently declared wells whose names match the name root will be processed. The wells are added to the list in the order in which they were first declared in the simulation.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Example

This example first creates a fresh well list *LIST1 containing 8 wells, opens these wells, adds 3 more wells (already open) to the list, then resets their oil rate target/limit. In addition, 4 wells are moved to *LIST2 and are shut.

```
WLIST
-- List  Oper  Well
-- Name  -atn  Names............
'*LIST1'  NEW  PR5 WEL63 VV6 TS53 PR22 VV12 TS7 TS19 /
'*LIST2'  MOV  WEL3 WEL5 VV2 TS3 /
/
WELOPEN
'*LIST1'  OPEN /
'*LIST2'  SHUT /
/
WLIST
-- List  Oper  Well
-- Name  -atn  Names............
'*LIST1'  ADD  WEL14 WEL12 WEL21 /
/
WELTARG
'*LIST1'  ORAT  5000 /
/
```
WLISARG  Resets operating targets/limits for a list of wells

This keyword can be used to reset a control or constraint value for a list of wells that has previously been set up with the keyword WLIST (or WLISTNAM). The operation of the WLISTARG keyword is similar to WELTARG, except that several wells can have their targets or limits reset to different values in a single data record. This allows the data to be presented in a more compact form than the alternative keyword WELTARG, which requires a separate line of data for each separate value. The facility is useful when simulating production history, where the target rates of all the wells have to be reset regularly.

Before WLISTARG can be used on a list of wells, the control data of each well must initially have been fully specified using WCONPROD or WCONINJE. If a well has been declared a history matching well (see keywords WCONHIST and WCONINJH) the WLISTARG keyword may be used to modify its BHP limit, VFP table number, and artificial lift quantity; the other quantities should not be modified with this keyword.

The WLISTARG keyword is followed by any number of data records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash.

1. The well list name (enclosed in quotes)
   The well list must previously have been set up with the keyword WLIST (or WLISTNAM).

2. Definition of the control or constraint quantity to be changed
   
   **ORAT**  Oil rate
   
   **WRAT**  Water rate
   
   **GRAT**  Gas rate
   
   **LRAT**  Liquid rate
   
   **CRAT**  Linearly combined rate (coefficients must be set with LINCOM)
   
   **RESV**  Reservoir fluid volume rate
   
   **BHP**  Bottom hole pressure
   
   **THP**  Tubing head pressure (A non-zero VFP table number must first have been set)
   
   **VFP**  VFP table number
   
   **LIFT**  Artificial lift quantity (A non-zero VFP table number must first have been set)
   
   **GUID**  Guide rate (For group controlled wells)

   **ECLIPSE 300 only**
   
   **WGRA**  Wet gas rate

   **ECLIPSE 300 only**
   
   **CVAL**  Calorific production rate

   **ECLIPSE 300 only**
   
   **REIN**  Reinjection fraction

   **ECLIPSE 300 only**
   
   **STRA**  Steam rate (Available only with the Thermal option)
ECLIPSE 300 only

**SATP** Saturation pressure offset (Available only with the Thermal option)

ECLIPSE 300 only

**SATT** Saturation temperature offset (Available only with the Thermal option)

3. 4,... New value of this quantity, for each well in the well list

   The first value applies to the first well in the list, the second value applies to the second well in the list, and so on.

   **UNITS:** The units depend upon the quantity chosen. The appropriate units are listed in the specification for keywords WCONPROD and WCONINJE.

   **DEFAULT:** If the quantity in item 2 is VFP, LIFT or GUID, then the values here may not be defaulted. For all other quantities, a defaulted value here will result in ECLIPSE using the corresponding flowing value for the well at the end of the previous timestep as the target or limit for the control or constraint in item 2.

   End data record with a slash (/).

   End the set of records with a blank record, containing just a slash.

**Example**

This example resets the oil rate target for the 8 wells on the list *PRLST1*, and resets the water rate target for the 4 wells on the list *INLST1*.

```
WLISTARG
-- List     Control   Value for
-- Name     Quantity   each well......
'*PRLST1'  ORAT      1500 2110 1930 1320 1670 1460 1290 1830 /
'*INLST1'  WRAT      3240 2970 2850 2740 /
/ 
```
Sets up a dynamic lists of well names

The keyword WLISTDYN may be used set up a dynamic well list. This list differs from a static, user-defined well list (see keyword WLIST) in that membership of the list is automatically reset at the end of each timestep. The membership is determined by a condition on a well type quantity. For example, the list might be made up of all the wells in a particular group that have a water cut greater than 0.7. Membership may also be defined as, for example, the 3 wells with the highest water cut belonging to a particular group.

Well lists whose names begin with an asterisk (*) may be entered wherever a well name root can be entered in the SCHEDULE section well keywords. Well lists may also be used in the ACTIONX keyword to define a grouping of wells, whose combined rates, totals or ratios can form part of a triggering condition in this keyword. Combined rates, totals and ratios for all the wells in a well list may also be written to the summary file using an appropriate group summary mnemonic followed by the well list name (see “Well and group flow quantities”).

The maximum number of dynamic well lists in a simulation is limited by item 12 of the WELLIMS keyword. A well may belong to more than one list at a time, up to a maximum number as set in item 11 of WELLIMS.

The WLISTDYN keyword should be followed by any number of data records, terminating with a blank record containing just a slash (/). Each data record consists of the following items, and is terminated with a slash. The data items are:

1. The well list name. This should consist of up to 8 characters, enclosed in quotes. The first character of the list name must be an asterisk and the second character must be a letter.

2. Group name or group name root

   A group name root, enclosed in quotes and ending with an asterisk (*), can be used to refer to several groups in one record.

   All the wells that belong to this group (or these groups) will be considered for membership of this dynamic well list. Note that wells which can be hierarchically traced to this group through intermediate groups will not be considered for membership of the dynamic well list.

3. Well mnemonic. This is the defining quantity for list membership.

   If the mnemonic refers to a production type quantity, then only production wells from the group defined in item 2 will be considered for membership of the list. Similarly if the mnemonic refers to an injection type quantity, only injection wells will be considered.

   The mnemonic must be one of:

   \- WOPR  Well oil production rate
   \- WOPT  Well oil production total
   \- WOIR  Well oil injection rate
   \- WOIT  Well oil injection total
   \- WWPR  Well water production rate
   \- WWPT  Well water production total
   \- WWIR  Well water injection rate
WWIT  Well water injection total
WGPR  Well gas production rate
WGPT  Well gas production total
WGIR  Well gas injection rate
WGIT  Well gas injection total
WLPR  Well liquid production rate
WLPT  Well liquid production total
WVPR  Well voidage production rate
WVPT  Well voidage production total
WVIR  Well voidage injection rate
WVIT  Well voidage injection total
WGOR  Well gas oil ratio
WWCT  Well water cut
WWGR  Well water gas ratio
WBHP  Well bottom hole pressure (production type quantity)
WTHP  Well tubing head pressure (production type quantity)

4. The operator for the membership condition. This must be one of:
   '>' (Greater than)
   '<' (Less than)
   'Highest'
   'Lowest'

5. The value for the membership condition.
   If the operator in item 4 is greater than or less than, then the unit of this value depends upon the quantity selected in item 3.
   If the operator in item 4 is highest or lowest, then this value is the number of wells selected from the top or bottom of a list ordered by the well quantity in item 3.

End the data record with a slash (/).
End the set of records with a blank record, containing just a slash.
Notes

- The automatic updating of a dynamic well list at the end of every timestep may be disabled by setting item 2 of this keyword to 'OFF' and leaving items 3 to 5 blank. The contents of this well list then become static. To reverse this action, set item 2 to 'ON' (and leave items 3 to 5 blank).

- If a dynamic well list name is entered in item 1 of a WLST keyword, then it becomes a static list and is no longer updated at the end of each timestep. Its contents are effectively frozen at this point and may be altered by the operations of the WLST keyword. Similarly, if a static well list name is entered in item 1 of this keyword, then the list is emptied and henceforth is defined by the membership condition in this keyword.

- The contents of all dynamic well lists are reported in the .PRT file at each report time.

Examples

Example 1

In this example a dynamic well list (*HIGHGOR) is created that contains all the wells that currently have a GOR exceeding 100 Mscf/stb. The ACTION keyword checks whether the combined oil production rate of all the wells in this well list falls below 50 stb/day. (Note that the condition on there being some oil production ensures that the action is not triggered for a list of empty wells.) When this is true, all the wells in this list are shut and the dynamic well list is disabled. This action is only performed once.

```plaintext
WLISTDYN
  '*HIGHGOR' ' ' WGOR > 100 /
/
ACTIONX
  ACT1 1 /
  GOPR '*HIGHGOR' > 0 AND /
  GOPR '*HIGHGOR' < 50 /
/
  WELOPEN
  '*HIGHGOR' SHUT /
/
WLISTDYN
  '*HIGHGOR' OFF /
/
ENDACTIO
```

Example 2

In this example a dynamic well list (*HIGHWCT) is created that contains the 3 wells from group GRP1 that currently have the highest water cut. These 3 wells then have their oil rate target reduced by 5%. The simulation is then advanced (and the dynamic well list automatically updated) and again the 3 highest water cut wells have their oil rate reduced by 5%.

```plaintext
WLISTDYN
  '*HIGHWCT' 'GRP1' 'WWCT' 'HIGHEST' 3 /
/
WTMULT
  '*HIGHWCT' ORAT 0.95 /
/
TSTEP
  100.0 /
WTMULT
  '*HIGHWCT' ORAT 0.95 /
/
TSTEP
  100.0 /
```
WLISNAM  Sets up lists of well names, for use in WLISTARG

The WLISTNAM keyword has been superseded by the WLIST keyword, which offers a better set of facilities for managing well lists. WLISTNAM is retained for back compatibility, and it must still be used to operate on any old-style list names (beginning with a letter), but these lists are not recognized in any keyword other than WLISTARG.

The keyword may be used to construct and maintain well lists, which are lists of well names. Well lists whose names begin with an asterisk (*) may be entered wherever a well name root can be entered in the SCHEDULE section well keywords, as an alternative method of specifying several wells at once. Well lists are also used by keyword WLISTARG to set individual operating targets or limits for several wells in a single data record.

Each list should have a unique name, consisting of up to eight characters. The first character should be an asterisk and the second character must be a letter. Alternatively, pre-2000A style list names (beginning with a letter) are allowed, but these lists are not recognized by any keyword other than WLISTARG. Each list may contain any number of well names. Up to 50 well names may be added to a list in a single data record. Lists containing more than 50 well names may be constructed by appending well names to the list in subsequent data records. Wells may be added to, deleted from or moved between lists at any time in the simulation. A well may belong to more than one list at a time, up to a maximum as set in item 11 of WELLDMIS.

The WLISTNAM keyword should be followed by any number of data records, terminating with a blank record containing just a slash (/). Each data record consists of the following items, and is terminated with a slash. The data items are

1  The well list name.
   This should consist of up to 8 characters. The first character of the list name should be an asterisk (*) and the second character should be a letter. Alternatively, pre-2000A style list names (beginning with a letter) are allowed, but these lists are not recognized by any keyword other than WLISTARG.
   If a well list with this name has already been declared, the well names entered in items 2 - 51 of this record are appended to the end of the list.

2 - 51  A list of up to 50 well names.
   The wells must previously have been declared with the WELSPECS keyword.
   Well name roots (ending with an asterisk) may also be used here. When a well name root is encountered, all currently declared wells whose names match the name root are processed. Wells are added to the list in the order in which they were first declared in the simulation.
   If the first well name (item 2) is a default (1*) or a blank (’’), all wells already on the list are first deleted from it, before any more wells are added. Thus the list is started anew, beginning with the well name in item 3.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Example

This example first adds 8 wells to the well list *PRLST1. It then deletes any existing wells from the list *INLST1 and starts it afresh with 4 wells:
-- List    Well
-- Name    Names............
'*PRLST1'  PR5  WEL63  VV6  TS53  PR22  VV12  TS7  TS19 /
'*INLST1'  1*   IW2   IW3  IW4  IW6 /
/
WMF

Specifies cell initial total water composition explicitly

This keyword specifies the initial total water composition values for each cell. The keyword should be followed by $N_x \cdot N_y \cdot N_z \cdot N_w$ values in normal order, with the X index changing fastest.

This keyword is normally used to specify the initial solution explicitly, as an alternative to equilibration using EQUIL. Explicit initialization is expected if the PRESSURE keyword is present. The syntax follows that of XMF and YMF.

WMF values that do not sum to 1.0 are normalized provided the error is less than 10%. An error greater than 10% is flagged as a data error.

See "Water components" in the ECLIPSE Technical Description for further details of the water component option.

Example

```plaintext
WMF
324*1.0
324*0.0
```

Keywords

2626
Total water composition with respect to depth tables

The data comprises a table of total composition with respect to depth data for each equilibration region. Each table consists of $N_w + 1$ columns of data, and is terminated with a slash (/).

Column:

1. Depth values.
   The values should increase monotonically down the column.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

2 to $N_w + 1$ The corresponding values of $z_i$, the total phase mole fractions. The $N_w$ mole fractions specified must add up to unity, and the program checks that this is the case.

For depth values outside the interval specified, constant value extrapolation is performed. If the composition is independent of depth, only a single row is required.

The default maximum number of rows per table is 50; this may be reset using the third item in keyword EQLDIMS. The number of equilibration regions is entered using the first item in keyword EQLDIMS.

See "Water components" in the ECLIPSE Technical Description for further details of the water component option.

Example

In this case a constant composition with respect to depth for 2 equilibration regions, with 3 water components:

```
WMFVD
1000.0 1.0 0.0 0.0
2000.0 0.0 1.0 0.0/
1000.0 1.0 0.0 0.0
2000.0 0.0 0.0 1.0 /
```

Keywords

2627
WNAMES

Water component names

In a run with \( N_w \) water components, this keyword associates an identifying character string with each component for reporting purposes. These may be of up to eight characters.

The keyword should be followed by \( N_w \) values. They may be enclosed in quotes, although these are only required when blank characters are used, or the name starts with a non-alphabetical character.

See "Water components" in the ECLIPSE Technical Description for further details.

Example

```
WNAMES
  Fresh Brine/
```
**WNETCTRL**  Selects well THP or rate limits to be set by network

This keyword can be used when running the network option (see "Network option" in the ECLIPSE Technical Description), to select the well control quantity that is held constant after the network has been balanced and until the next balancing calculation. Normally this quantity is the well THP limit, which is fixed equal to its group’s nodal pressure in the network (plus any fixed pressure drop specified in keyword WNETDP). By default, this will be the network control quantity for all the wells if the WNETCTRL keyword is absent.

Consider, for example, the gas flow rate selected as the control quantity. The network will be balanced as usual by setting the wells’ THP limits from their group’s nodal pressure. However, any wells constrained by the network (that is that are under THP control in the balanced network) will have their gas flow rate limit fixed at its current value until the next network balancing calculation, and their THP limit removed. Any gas flow rate limit set by the user will still be honored, if ever that becomes the acting constraint.

It is expected that the default control quantity of THP will be the most suitable one for most simulations. Circumstances where it may be worth considering the flow rate as a control quantity are high-permeability fields with a low drawdown at the wells, which are coupled to a network in which the pressure losses are strongly sensitive to the flow rate. This may be the case in high-permeability gas reservoirs with compressors in the network. Keeping the well THP fixed until the next balancing calculation would generate large errors in the well flow rate if the change in reservoir pressure over the timestep is a significant fraction of the drawdown.

In general, this keyword should only be used if problems arise with THP as the control quantity; for example if large network balancing errors are reported at the end of the timestep. If large network balancing errors are reported, first check the NETBALAN keyword to ensure the convergence criteria are suitable and that the network is being balanced at every timestep. If this is so, then try setting a negative balancing interval to make ECLIPSE balance the network in each of the first NUPCOL Newton iterations of the timestep. If problems persist, or if the Newton iteration convergence is badly affected, then try selecting the flow rate to be the network control quantity.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. **Well name, well name template, well list or well list template**
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. **Control quantity to remain fixed after each network balancing calculation when the well is limited by the network**
   
   THP  Tubing head pressure
   
   LRAT Liquid rate (Production wells only)
   
   GRAT Gas rate
   
   WRAT Water rate (Water injection wells only)
   
   **DEFAULT:** THP
End the data record with a slash (/).
End the set of records with a blank record, containing just a slash.

Example

```
WNCTRL
 'PROD*' THP /
 'GINJ*' GRAT /
 / 
```
Fixed pressure drops between network node and THP

This keyword can be used when running the network option (see "Network option" in the ECLIPSE Technical Description), to apply an additional fixed pressure drop between a well's tubing head and its group's corresponding node in the network. Normally the wells' THP limits are set equal to their group's nodal pressure at each network balancing calculation. But if a fixed pressure drop is specified with this keyword for a well, the well's THP is set as follows:

- for a production well in a production network:
  well THP = group’s nodal pressure + fixed pressure drop for well
- for an injection well in an injection network:
  well THP = group’s nodal pressure - fixed pressure drop for well.

The keyword can also be used to isolate individual wells from the effects of the network. If the fixed pressure drop is set to a value greater than or equal to 1.0E10, the well's THP limit remains unchanged by the network balancing calculation and is not set equal to its group's nodal pressure. The well's flow, however, is still added to the network flows.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Fixed pressure drop between the well and its group’s network node

   A value greater than or equal to 1.0E10 causes the well's THP limit to become independent of its group's nodal pressure. It remains constant unless changed manually in, for example, keyword WCONPROD.

   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

   DEFAULT: 0.0

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Example

```
WNETDP
PROD1  20.0 /
PROD2  30.0 /
```

Keywords

WebNet
**WORKLIM**  Time taken for each automatic workover

The keyword should be followed by a single real number that represents the time taken to do each workover, followed by a slash (/). This imposes a limit on the number of well workovers that can be performed by a single workover rig in a given timestep. The limit is equal to the timestep size divided by the time taken for each workover. In ECLIPSE 100, a workover refers to the automatic closure, plug-back or opening of a well connection resulting from instructions specified in keywords **WECON, CECON, GECON, GCONPROD, GCONPRI, GCONSALE, PRORDER** and **WORKTHP**. In ECLIPSE 300, a workover currently refers only to the automatic closure of a well connection as a result of an economic or rate limit being broken or a **WORKTHP** event. Connection changes performed manually (for example using **COMPDAT**) are not affected.

**UNITS:** day (**METRIC**), day (**FIELD**), hr (**LAB**), day (**PVT-M**)

When several workovers need to be performed at any one time, they are performed in the following order of priority:

1. Workovers arising when a well cannot produce at its THP limit
2. Workovers arising from connection economic limit violations
3. Workovers arising from well economic limit violations
4. Workovers arising from group economic limit violations
5. Workovers arising from group flow limit violations
6. Workovers arising from field economic limit violations
7. Workovers arising from field flow limit violations.

Within each category, workovers are performed in decreasing order of fractional violation (for example, actual GOR / limiting GOR).

If this keyword is absent the workover duration is defaulted to zero, and there is no limit on the rate of doing workovers.

Wells that are being worked over may be temporarily closed by setting item 3 in the keyword **WDRILTIM** to ‘YES’.

See also the keyword **GRUPRIG** which assigns workover rigs to groups.

**Example**

```
WORKLIM
5.0 /
```

---

**Keywords**

2632
**WORKTHP**  Workover procedure if well dies under THP control

The keyword may be used to instruct ECLIPSE to perform a workover if a well is unable to produce at its THP limit, to reduce its water cut. The workover closes the connection that had the highest water cut, and optionally any connections below it, or plugs back the bottom of the perforations. The workover is performed at the end of the timestep in which the well dies, and the well is revived at the beginning of the next timestep to see if it can produce at the reduced water cut. If the well still cannot produce, workovers are performed at the end of each timestep until all the connections are closed.

If the economic limit tolerance fraction has been set less than 1.0 in keyword **WLIMTOL**, the timestep will be repeated after the workover has been performed. If connections are lumped together into completions (see keyword **COMPLUMP**), the workover closes all connections in the worst-offending completion (and optionally all below). If, in ECLIPSE 100, the well has any connections in its automatic opening queue (see item 6 of keyword **COMPDAT**) and which meet the minimum oil saturation requirement (if imposed with keyword **WELSONMIN**), one of these is opened if any connection is closed during a workover. See also keywords **WORKLIM** and **GRUPRIG** for limiting the rate at which workovers can be performed.

The **WORKTHP** keyword is followed by any number of records, each containing the following two items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   - A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword **WLIST**.

2. Workover procedure if the well is unable to produce at its THP limit
   - **NONE**  No workover, let the well die
   - **CON**   Shut connection with highest water cut
   - **+CON**  Shut connection with highest water cut, and all below it (see keyword **COMPORD**)
   - **PLUG**  Plug back the well (see keyword **WPLUG**)

   *ECLIPSE 100 only*

   **DEFAULT:** **NONE**

End data record with a slash (/).

End the set of records with a blank record, containing just a slash (/).

**Example**

```
WORKTHP
  'P*'  CON /
  PROD2  NONE /
/  
```

---

**Keywords**

2633
WPAVE

Well block average pressure controls

This keyword controls the calculation of well block average pressures. These averages represent the average pressure of the grid blocks containing connections to a given well, and optionally their adjacent and diagonal neighbors also, weighted according to either the connection transmissibility factors or the grid block pore volumes. WPAVE controls the averaging method for all wells in the simulation. Alternatively, pressure averaging controls for individual wells can be specified using the WWPAVE keyword. WPAVE overwrites any current controls specified by the WWPAVE keyword.

The average pressures can be written to the Summary file using the SUMMARY section keywords WBP, WBP4, WBP5 and WBP9. Well PIs based on these average pressures can also be written to the Summary file using the SUMMARY section keywords WPI1, WPI4, WPI5 and WPI9 (see "SUMMARY section").

The averages pertain to a Cartesian grid and should not be used in a radial grid. The averages are used for reporting purposes only, and will not affect any other results.

The keyword should be followed by a line containing the following items of data, terminated by a slash (/):

1. F1, the weighting factor between the inner block and the outer ring of neighboring blocks, in the connection factor weighted average.

   If the value lies between 0.0 and 1.0, the average is calculated by equation 3.236. The value 1.0 gives total weighting to the inner blocks, containing the well connections. The value 0.0 gives total weighting to the 4 or 8 blocks neighboring each inner block.

   A value of F1 < 0.0 is used to indicate that the pressure of the inner block and its outer ring of neighboring blocks should be averaged according to their pore volumes. The average block pressure for each connection is calculated by equation 3.237.

   DEFAULT: 0.5

   Note: For one-block averages (WBP) F1 is effectively 1.0, whatever the value entered here.

2. F2, the weighting factor between the connection factor weighted average and the pore volume weighted average, used by equation 3.234.

   The value should lie between 0.0 and 1.0. The value 1.0 gives a purely connection factor weighted average, and 0.0 gives a purely pore volume weighted average.

   DEFAULT: 1.0

3. Depth correction flag

   This flag controls how the grid block pressures are corrected to the well’s bottom hole reference depth (see item 5 in keyword WELSPECS), or to an alternative reference depth if specified in keyword WPAVEDEP, by applying a hydrostatic head between each grid block center and the well’s reference depth.

   WELL The hydrostatic head is calculated using the density of the fluid in the wellbore at the well connections.

   RES The hydrostatic head is calculated using a representative density for the fluid in the reservoir. This density is calculated by averaging over fluid density for all the grid blocks associated with the well’s connections. The average over phases is weighted by the phase saturation, and the average over grid blocks is weighted by pore volume. Whether the averaging is performed
over all grid blocks with declared connections to the well or only those with currently open connections is determined by item 4.

NONE  Grid block pressures are not depth corrected.

DEFAULT: WELl

Note: The wellbore fluid density is set to zero whenever the well is shut. Thus if WELl is selected there is a discontinuity in the reported pressure average when the well’s status changes between shut and open/stopped.

4. Well connection flag

OPEN  Only grid blocks associated with currently open connections are included in the averaging calculation.

ALL  Grid blocks associated with all currently defined connections (whether open or closed) are included in the averaging calculation.

DEFAULT: OPEN

Note: If OPEN is selected there is a discontinuity in the reported pressure average whenever new connections are opened or existing ones are closed. This may be avoided by selecting ALL and defining all the well’s connections at the start of the run (whether initially open or closed).

If this keyword is not present, all items assume their default values, giving a connection factor weighted average, evenly weighted between the inner blocks and the outer ring of neighbors. The depth correction uses the wellbore density and only grid blocks associated with currently open well connections are included in the average.

Note: In dual porosity runs the wells are completed in the fracture system. A WBPn average reports the average pressure in the surrounding fracture cells.

Examples

Example 1
To give a connection factor weighted average, with increased weighting to the outer blocks:

WPAVE 0.2 1.0 /

Example 2
To give a purely pore volume weighted average over all the blocks containing open or closed well connections and their neighbors:

WPAVE 1* 0.0 1* ALL /
Example 3

To give a connection factor weighted average, where the pressure for each connection is the pore volume weighted average of pressures in the connection block and its outer ring of neighbors. The grid block pressures are not depth corrected:

```
WPAVE -1.0 1.0 NONE /
```

Calculation of well block average pressures

The well block average pressure $P_w$ for a given well is a weighted combination of the connection factor weighted average pressure $P_{w,cf}$ and the pore volume weighted average pressure $P_{w,pv}$

$$P_w = F2 P_{w,cf} + (1-F2) P_{w,pv}$$

Eq. 3.234

Connection factor weighted average

This is the average over connections of the average block pressure $P_k$ at each (open or declared, depending on item 4) connection $k$, weighted according to the connection transmissibility factors $T_k$

$$P_{w,cf} = \frac{\sum_k T_k P_k}{\sum_k T_k}$$

Eq. 3.235

When $F1 \geq 0$, the average block pressure for each connection $k$ is the weighted average of the inner block pressure $P_{i,k}$ (that is the block containing the connection) and the average of the pressures in the 4 or 8 blocks surrounding it $P_{o,k}$

$$P_k = F1 P_{i,k} + (1-F1) \frac{\sum_{o,k} P_{o,k}}{N_{o,k}}$$

Eq. 3.236

when $F1 \geq 0$

When $F1 < 0$, the average block pressure for each connection $k$ is the average of the pressures in the inner block $P_{i,k}$ and in each of the 4 or 8 blocks surrounding it $P_{o,k}$, weighted according to their pore volumes $V_{i,k}$ and $V_{o,k}$

$$P_k = \frac{V_{i,k} P_{i,k} + \sum_{o,k} V_{o,k} P_{o,k}}{V_{i,k} + \sum_{o,k} V_{o,k}}$$

Eq. 3.237

when $F1 < 0$

The pressure in each individual grid block $P_{i,k}$ or $P_{o,k}$ is corrected to the well’s bottom hole reference depth according to the option selected in item 3, or to an alternative reference depth if specified in keyword WPAVEDEP.

The number of surrounding blocks $N_{o,k}$ is 4 for four-block and five-block averages (WBP4, WBP5) and 8 for nine-block averages (WBP9). The configuration is shown in figure 3.18. Four- and five-block averages use the four immediate neighbors (n) of the connecting grid block. Nine-block averages use in addition the four diagonal neighbors (d). The inner block is ignored in four-block averages (WBP4). The number of neighbors is smaller if the well is situated on the edge of the grid or adjacent to an inactive cell. The
neighbors are selected in the plane perpendicular to the direction of penetration of the connection (see item 13 of keyword COMPDAT). Thus for horizontal wells, the neighbors are in a vertical plane.

**Pore volume weighted average**

This is simply the average depth-corrected pressure $P_j$ in the selected set of grid blocks $j$, weighted by their pore volumes $V_j$

$$P_{w, pv} = \frac{\sum j V_j P_j}{\sum j V_j} \quad \text{Eq. 3.238}$$

The set of grid blocks $j$ is selected from the blocks containing (open or declared, depending on item 4) connections of the well (i), their four immediate neighbors (n), and their four diagonal neighbors (d), in a similar manner to that described above for the other averaging scheme. One-block averages ($WBP$) just use the inner blocks (i). Four-block averages ($WBP4$) just use the immediate neighbors (n). Five-block averages ($WBP5$) use both (i) and (n). Nine-block averages ($WBP9$) use (i), (n) and (d).

<table>
<thead>
<tr>
<th></th>
<th>d</th>
<th>n</th>
<th>d</th>
</tr>
</thead>
<tbody>
<tr>
<td>n</td>
<td>i</td>
<td>n</td>
<td></td>
</tr>
<tr>
<td>d</td>
<td>n</td>
<td>d</td>
<td></td>
</tr>
</tbody>
</table>

*Figure 3.18. Grid block configuration in well block average pressure calculations*
WPAVEDEP  Reference depth for well block average pressure calculation

This keyword may be used to modify the reference depth for the calculation of well block average pressures (see keyword WPAVE) for output to the Summary file. By default, the grid block pressures are corrected to the well’s bottom hole reference depth (see item 5 in keyword WELSPECS) by applying a hydrostatic head. The WPAVEDEP keyword enables the use of an alternative reference depth, which is used solely for this calculation.

The depth correction is calculated using either the fluid density in the well bore at the depth of the well connections, or the fluid density in the grid blocks associated with the well’s connections, as requested in item 3 of keyword WPAVE.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. The reference depth for the calculation of well block average pressures.

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

   DEFAULT: Use the well’s bottom hole reference depth

   End data record with a slash (/).

   End the set of records with a blank record, containing just a slash (/).

Example

```
WPAVEDEP
   'PROD1*'  3640 /
/
```

Keywords
This keyword can be used to multiply the connection transmissibility factors of selected well connections by a specified value. To multiply the transmissibility factors of all the connections in a well, leave items 3 - 7 defaulted. To multiply the transmissibility factors of a subset of connections in a well, you can identify the subset by their I,J,K location (items 3 - 5). A subset of connections can also be identified by their completion number (items 6 and 7). Completion numbers are set in keyword COMPLUMP.

The WPIMULT keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template.
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Multiplier to act on the well’s connection transmissibility factors and Kh values.
   The value must be greater than 1.0E-10
   DEFAULT: 1.0

3. I - location of connecting grid block(s)
   DEFAULT: Negative (allows any I-location)

4. J - location of connecting grid block(s)
   DEFAULT: Negative (allows any J-location)

5. K - location of connecting grid block(s)
   DEFAULT: Negative (allows any K-location)

6. Number of first completion in range
   DEFAULT: Negative; allows any completion number up to item 7 (if set)

7. Number of last completion in range
   DEFAULT: Negative; allows any completion number from item 6 (if set)

If items 3 - 7 are all defaulted, the connection factor multiplier applies to all connections in the well.

If any of items 3 - 7 are set positive, the multiplying factor applies only to a subset of the connections in the well. The connections in the subset are those with I,J,K locations matching the specified location indices in items 3 - 5 and a completion number in the range specified by items 6 - 7. A zero or negative value for a location index (I, J or K) matches any value of that location index. Similarly a zero or negative value for a completion range lower/upper end-point indicates that there is no lower/upper bound on the completion numbers.
Notes

• When WPIMULT is read, the connection factors of the chosen set of connections are immediately multiplied by the specified factor. If the COMPDAT keyword is subsequently used to reset any of their connection data, this causes the connection factors to be reset. If the multiplier is still required, WPIMULT must be entered again after COMPDAT.

• If WPIMULT is used more than once on the same set of well connections, without resetting the connection factors with an intermediate COMPDAT keyword, its effect is cumulative if there are intervening timestepping keywords - that is, the multiplying factor is applied to the chosen set of connections each time the keyword is entered.

• If the keyword applies to all the connections in a well (that is, the connection and completion items are defaulted) and it is used more than once, its effect will only be cumulative if at least one timestepping keyword falls between the WPIMULT (WPIMULTL) keyword repeats. If this is not the case and several keywords are defined for all the connections in a particular well before a timestepping keyword, only the last defined multiplier will be applied to the connection factors before the simulation is advanced.

• Keywords WELPI and WPIMULT should not both be used on the same well at the same time, that is without an intervening TIME, TSTEP or DATES keyword.

Note: This keyword can be used with OPTIONS3 (item 43) to avoid an error message being generated when a local grid well is referred to.

Example

```
WPIMULT
  PROD1  1.3  /(all connections in the well PROD1)
  PROD2  0.82 1* 1* 5  /(all connections of well PROD2 in layer 5)
  PROD3  0.6 1* 1* 1* 7 7  /(all connections of PROD3 in completion number 7)
/
```
WPIMULTL  Multiplies well connection factors by a given value within local grids

This keyword can be used to multiply the connection transmissibility factors of selected local grid well connections by a specified value. To multiply the transmissibility factors of all the connections in a well, leave items 3 - 8 defaulted. To multiply the transmissibility factors of a subset of connections in a well, you can identify the subset by their local grid name and I,J,K location (items 3 - 6). A subset of connections can also be identified by their completion number (items 7 and 8). Completion numbers are set in keyword COMPLMPL.

WPIMULTL performs the same role as WPIMULT, but the name of the local grid containing the connections specified in the record should be entered in item 3 of WPIMULTL. This keyword should be used when the well is completed in a local grid.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Multiplier to act on the well’s connection transmissibility factors and Kh values
   The value must be greater than 1.0E-10
   DEFAULT: 1.0

3. Name of local grid containing the connections in items 4 - 6
   If this item is defaulted or set to a blank string, the local grid name is set to the well’s local grid named in keyword WELSPECL. (In ECLIPSE 100, the local grid name should not be defaulted if the local grid is amalgamated.)
   The local grid name may also be defaulted if the connection locations in items 4, 5 and 6 are all defaulted. The multiplier in item 2 then applies to all the connections in the well (or to specific completions within it if items 7 and 8 are set).
   DEFAULT: ''

4. I - location of connecting grid block(s)
   DEFAULT: Negative (allows any I-location)

5. J - location of connecting grid block(s)
   DEFAULT: Negative (allows any J-location)

6. K - location of connecting grid block(s)
   DEFAULT: Negative (allows any K-location)

7. Number of first completion in range
   DEFAULT: Negative; allows any completion number up to item 8 (if set)
8. Number of last completion in range

   DEFAULT: Negative; allows any completion number from item 7 (if set)

If items 4 - 8 are all defaulted, the connection factor multiplier is applied to all connections in the well (or all connections in the specified local grid if a local grid name is entered in item 3).

If any of items 4 - 8 are set positive, the multiplying factor applies only to a subset of the connections in the well or local grid. The connections in the subset are those with I,J,K locations matching the specified location indices in items 4 - 6 in the local grid named in item 3, and with a completion number in the range specified by items 7 - 8. A zero or negative value for a location index (I, J or K) matches any value of that location index. Similarly a zero or negative value for a completion range lower/upper end-point indicates that there is no lower/upper bound on the completion numbers.

Notes

- When WPIMULT (L) (WPIMULTL) is read, the connection factors of the chosen set of connections are immediately multiplied by the specified factor. If the COMPDAT (COMPDATL) keyword is subsequently used to reset any of their connection data, this causes the connection factors to be reset. If the multiplier is still required, WPIMULT (WPIMULTL) must be entered again after COMPDAT(COMPDATL).

- If WPIMULT(WPIMULTL) is used more than once on the same well, without resetting the connection factors with an intermediate COMPDAT(L) keyword, its effect is cumulative - the multiplying factor is applied to the chosen set of connections each time the keyword is entered.

- If the keyword applies to all the connections in a well (that is, the connection and completion items are defaulted) and it is used more than once, its effect will only be cumulative if at least one timestepping keyword falls between the WPIMULT(WPIMULTL) keyword repeats. If this is not the case and several keywords are defined for all the connections in a particular well before a timestepping keyword, only the last defined multiplier will be applied to the connection factors before the simulation is advanced.

- Keywords WELPI and WPIMULT(WPIMULTL) should not both be used on the same well at the same time, without an intervening TIME, TSTEP or DATES keyword.

Example

```plaintext
WPIMULTL
  PROD1 1.3 / (all connections in the well PROD1)
  PROD2 0.82 LGR2 / (all connections that the well PROD2 has in LGR 'LGR2')
  PROD3 0.98 4* 5 8 / (applies to completions 5,6,7 and 8 of well PROD3)
  PROD4 0.5 LGREAST 2* 3 / (all connections in layer 3 of LGR 'LGREAST')
  PROD5 0.75 LGRWEST 3 19 7 / (Only applies to connection (3,19,7) of LGR 'LGRWEST')
```

Keywords

2642
WPITAB

Assigns PI multiplier tables to wells

This keyword may be used to assign PI multiplier tables to individual wells. These tables scale the well’s connection factors according to the maximum water cut it has so far achieved, and are input with the PIMULTAB keyword.

The WPITAB keyword is followed by any number of lines, each containing the following data and each terminated with a slash (/). The last line must be followed by a line containing only a slash.

1. Well name, well name template, well list or well list template.
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. PI multiplier table number.
   An integer no greater than NTPIMT (see keyword PIMTDIMS in the RUNSPEC section).
   A value of 0, or a negative value, will mean that no PI multiplier table will be assigned to that well.
   DEFAULT: 0

See also keyword WPIMULT, which may be used to multiply well PIs by a constant factor. Alternatively, keyword WSCTAB and its associated keywords may be used to modify the PI dynamically according to a scale deposition model. WPITAB and WSCTAB must not both be used on the same well.

Example

```plaintext
WPITAB
  'P*' 2 /
  PROD2 1 /
  PROD3 0 /
```

Keywords

2643
WPLUG
Sets well plug-back lengths

This keyword must be used to set the plug-back lengths for all wells that may be subject to automatic plug-backs (requested in keywords WECON, GECON, GCONPROD, GCONPRI, GCONSALE, WECONT, GECONT and WORKTHP).

When a well is subject to an automatic plug-back a specified length of perforations is closed, starting from the top or bottom of the open connections in the well, or the top or bottom of the open connections in the worst offending completion, depending on the entry in item 4. The decision on whether to start from the top or the bottom depends on the reason for the plug-back; plug-backs to reduce the gas fraction starts from the top while those intended to reduce the water fraction will start from the bottom. A plug-back will not necessarily close an entire connection. A connection is only closed when there is no length of perforations remaining open in the grid block.

A plug-back will alter the depth of the top or bottom of the open perforations, thus affecting the ratio of fluids produced by the well. The initial top and bottom depths of the perforated length may be set in keyword COMPVE, or defaulted to the center depths of the top and bottom grid block faces to give full penetration of the grid block. If item 10 in COMPVE is set to YES, the skin factor and transmissibility factor of the connection will also be recalculated after each plug-back, to reflect the reduced length of open perforations.

When plug-backs occur in runs that do not use the Vertical Equilibrium option, the new perforation depths are automatically converted into scaled saturation table endpoints, which are then treated as if they had been entered with the COMPFP keyword. The conversion is performed by assuming a VE-type segregation of phases within the grid block (see "Representing the effects of partial penetration"). There is a restriction on using plug-backs in non-VE runs that are fast restarts (see keyword RESTART). The required grid block geometry data is only stored on the SAVE file if the base run contained the keyword COMPVE. So if you wish to use plug-backs during a subsequent fast restart, you should include keyword COMPVE in your base run, followed by just a slash if you do not need to set any partial penetration data at that time. The restriction does not apply to flexible restarts, as the required data are generated from the GRID section.

The WPLUG keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Length of perforation closed each time there is a plug-back from the top (for example for high GOR)
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   This item must be set to a value greater than zero.

3. Length of perforation closed each time there is a plug-back from the bottom (for example for high water cut)
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   This item must be set to a value greater than zero.

4. Perform plug-backs from the top/bottom of the well or the worst offending completion?
WELL  Plug-backs to be performed from the top/bottom of the open connections in the well.

COMP  Plug-backs to be performed from the top/bottom of the worst offending completion.
       (Connections are lumped into completions with keyword COMPLUMP.)

DEFAULT: WELL

Only the first letter is significant.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Example

```
WPLUG
 'P*' 5.0 5.0 /
 PROD2 2.0 3.0 COMP /
 /
```
W POLYMER  Sets polymer/salt concentrations for injection wells

This keyword is used to specify the concentration of polymer and salt in the injection stream of each well. If the W POL YMER keyword does not appear in the injection schedule, then concentration values of zero are assumed. Data should only appear for wells which are currently declared to be water injection wells.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template.
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword W LIST.

2. The concentration of polymer in the injection stream for the well. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.
   UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm$^3$ (PVT-M)

3. The concentration of salt in the injection stream for the well. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.
   If the salt-sensitivity option is not activated (using keyword BRINE in the RUNSPEC section), then the salt concentration entry is ignored.
   UNITS: kg/sm$^3$ (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm$^3$ (PVT-M)

4. The group whose produced polymer concentration should be used for the polymer concentration of the injection stream for the well.
   If no group specified the concentration in item 2 will be used

5. The group whose produced salt concentration should be used for the salt concentration of the injection stream for the well.
   If no group specified the concentration in item 3 will be used

End data record with a slash (/).
End the set of records with a blank record, containing just a slash (/).

Example

```
W POLYMER
INJ1 0.3 1.0 /
INJ2 0.0 0.0 /
INJ3 0.3 1.0 /
```

Keywords

2646
WPOLYRED  Sets polymer/water mixture viscosity in wells

This keyword is used to specify the viscosity reduction factor, \( \alpha \), for polymer/water mixtures in wells. If this value is set to 1.0, the water phase viscosity in the well is calculated using the mixture viscosity set out in the Treatment of fluid viscosities in the well in the ECLIPSE Technical Description. If the viscosity reduction factor is set to zero, the pure water viscosity is used. A value between 0.0 and 1.0 gives an interpolated viscosity between these two values.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template.
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. The polymer viscosity reduction factor.
   
   UNITS: dimensionless
   
   DEFAULT: 1.0

   End data record with a slash (/).

   End the set of records with a slash (/).

**Example**

```
WPOLYRED
  INJ1 0.3 /
  INJ2 0.0 /
  INJ3 0.3 /
/
```
WPOTCALC  Control calculation of shut and stopped well potentials

This keyword controls whether shut and stopped well potentials are always calculated, or only when necessary. By default, since 2007.1, these calculations are always performed.

If you do not require these potentials, then the calculation can be turned off. An example of a situation where the calculation of shut and stopped well potentials is not required is when a stopped well is segmented all the way to the surface (for example to look at the phase segregation process after the well is stopped). Such a well cannot operate at a constant THP and so the shut well potential calculation will not converge.

Shut and stopped well calculations are necessary in the following situations:

• When the well is in a priority drilling queue (see WDRILPRI) or when the prioritization option is used (see PRIORITY).

• When any group, well or connection requests a potential summary vector, for example using the FOPP keyword in the SUMMARY section.

• When an action uses a potential (see ACTIONX).

The calculation of shut and stopped well potentials will be performed regardless of the WPOTCALC setting in these necessary situations.

The keyword is followed by one item of data, terminated with a slash (/).

1. Flag indicating whether well potential calculations should be forced for all wells, regardless of whether they are shut or stopped.

   NO  Shut and stopped well potentials are not calculated unless necessary.

   YES Shut and stopped well potentials are always calculated.

   (The word may be abbreviated; only the first character is significant)

   DEFAULT: YES

End the data record with a slash (/).

Note: Turning off the additional calculations of well potentials using this keyword is equivalent to specifying the item 101 in OPTIONS to a non-zero multiple of 3.

Example

WPOTCALC
NO /
WREGROUP  Automatic regrouping of wells

This keyword can be used to transfer wells automatically from one group to another, depending on the value of a chosen quantity such as the well’s water cut or oil production rate. Some examples of where this may be useful are given below:

- There are two separate production streams, one for wells cutting significant amounts of water and one for wells with very low water cut. Wells are transferred from one stream to the other depending on their water cut.

- Wells can be classified as either high pressure wells or low pressure wells, depending on the calculated values of their tubing head pressure when they are flowing at a controlled rate.

- In runs using the Network option, wells can be moved from a high pressure network node to a low pressure network node to boost their production, once their original flow rate falls below a specified value.

Two limits are provided, an upper one and a lower one, so that the transfer between groups can be either one way or reversible. The limit values need not be the same for each direction, so that a reversible transfer may have a degree of hysteresis. The transfers take place at the start of the next timestep after a limit has been passed. The transfers take place instantaneously with no loss of production, and there is no restriction on the rate at which transfers can be performed.

**Note:** Further wells are not transferred into a group if they would exceed the maximum number of open wells in the group or a superior group, set in item 9 of keyword GECON.

**Note:** If setting the well dimensioning data manually in the RUNSPEC section keyword WELLDIMS, the value of NWGMAX (the maximum number of wells in any one group) must allow for all possible transfers.

The WREGROUP keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The records may be terminated at any item after item 1. The remaining items will assume their default values. Default values can be specified before the slash by a null repeat count of the form n*, where n is the number of consecutive items to be defaulted.

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Name of the quantity on which the switching will be decided
   
   WCT  Water cut
   
   GOR  Gas-oil ratio
   
   WGR  Water-gas ratio
   
   ORAT  Oil rate

Keywords

2649
GRAT  Gas rate
THP  Tubing head pressure
  ' '  No regrouping for this well
DEFAULT: ' '

3. Name of the group into which the well should be moved when its specified quantity exceeds the value in item 4
   DEFAULT: The group in which the well was placed using the WELSPECS keyword

4. Value of the quantity specified in item 2 above which the well is moved into the group specified in item 3
   The value must not be less than the value of item 6
   DEFAULT: Infinity (1.0E20), which prevents switching on this condition

5. Name of the group into which the well should be moved when its specified quantity falls below the value in item 6
   DEFAULT: The group in which the well was placed using the WELSPECS keyword

6. Value of the quantity specified in item 2 below which the well is moved into the group specified in item 5
   The value must not be greater than the value of item 4
   DEFAULT: 0.0, which prevents switching on this condition

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Example

In this example all wells whose names start with PROD are moved into group WET when their water cut exceeds 0.2, and are moved back to their original group if their water cut subsequently falls below 0.1. That is except for well PROD3, which stays in the group specified in WELSPECS regardless of its water cut.

WREGROUP
  'PROD*'  WCT  WET  0.2  1*  0.1 /
  PROD3  /
  /
Request output to RFT file

This keyword initiates output of well RFT data to the RFT file. The data consists of the pressure, saturation and depth values for each grid block in which a well has a connection. The keyword is retained for back compatibility; however, you are encouraged to use keyword **WRFTPLT**, which provides a more comprehensive set of options for writing well RFT and PLT data to the RFT file.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template.
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword **WLIST**.

   End the data record with a slash (/).

   End the set of records with a blank record, containing only a slash (/).

   If the keyword is entered without any data records (that is followed only by a blank record), well RFT data will subsequently be output whenever a well is first opened. If the keyword contains data records referring to well names, then in addition the RFT data for those wells are output at that time in the simulation.

Examples

**Example 1**

Request RFT data output for any well when it is first opened.

```plaintext
WRFT
/
```

**Example 2**

Request current RFT data output for specified wells, plus output when any well is subsequently opened.

```plaintext
WRFT
  PROD1 /
  'A*'  /
```
Requests output of well RFT, PLT and segment data to the RFT file

The keyword initiates output of well data to the RFT file. The well must previously have been introduced in the **SCHEDULE** section. This file contains data for one or more wells describing fluid conditions in the wellbore or the connecting grid blocks at selected times in the run. There are three categories of data that may be written to this file:

RFT data for grid blocks containing connections to the well:

- Depth
- Pressure
- Water and gas saturations
- Polymer and brine concentrations

PLT data for each connection in the well:

- Depth
- Pressure
- Oil, water, gas, polymer and brine flows
- Polymer and brine concentrations
- Tubing flows at each connection (the total upstream flow rates at both surface conditions and local wellbore conditions)
- Connection transmissibility factor and Kh
- Length down the tubing to the connection (ECLIPSE 100 friction wells and multisegment wells only)
- Segment number containing the connection and its branch number (multisegment wells only)

Segment data for the multisegment well model (see "Multisegment wells" in the *ECLIPSE Technical Description*):

- Depth
- Tubing length
- Pressure
- Oil, water, gas, polymer and brine flow rates
- Oil, water and gas velocities
- Oil, water and gas holdup fractions
- Polymer and brine concentrations
- Oil, water, gas and polymer/water mixture viscosities
- Saturation pressure if **SPSAT** is present in summary section (ECLIPSE 300 only)
- Temperature (ECLIPSE 300 THERMAL only)
- Steam quality (ECLIPSE 300 THERMAL only)
- Enthalpy flow rate (ECLIPSE 300 THERMAL only)

**Keywords**
• Energy density (ECLIPSE 300 THERMAL only)
• Information required to construct multilateral well structures including branch data and pointers to the neighboring downstream segment
• Setting (fraction open) of ICD on segment
• Base strength of ICD on segment

River data (ECLIPSE 100 only)
• Reach length
• Flux
• Area
• Reach water depth
• Exchange flux with aquifer
• Froude number
• Information required to construct river structures including branch data, branch name and pointers to the neighboring downstream reach

The RFT file can be read by GRAF. Alternatively it may be examined visually if written as formatted output (see keyword `FMTOUT`). Unformatted RFT files can be converted to formatted files by using the CONVERT utility. See “File handling in ECLIPSE” in the ECLIPSE Technical Description. A detailed list of the contents of the file can be found in the ECLIPSE File Formats Reference Manual.

The connection ordering of the relevant RFT and PLT data is controlled by keyword `COMPORD` for standard wells, and always follows the `TRACK` method for multisegment and friction wells.

The `WRFTPLT` keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword `WLIST`.

   Note that this well name, well name template, well list or well list template must have already been defined in the `SCHEDULE` section.

2. Output RFT data?

   YES Output the current RFT data for the well(s) at this time, describing conditions in the grid blocks containing well connections.

   REPT Output the current RFT data for the well(s) at this time and at all subsequent report times.

   TIMESTEP Output the current RFT data for the well(s) at this time and at all subsequent timesteps.
FOPN  Output the current RFT data for the well(s) at this time if they are open. If any of the
wells named in item 1 are not yet open, output their RFT data subsequently when they
are first opened.

NO    Do not output RFT data for the well(s).

DEFAULT:  NO

3. Output PLT data?

YES   Output the current PLT data for the well(s) at this time, describing the flows through
the well connections.

REPT  Output the current PLT data for the well(s) at this time and at all subsequent report
times, while they are open or stopped.

ECLIPSE 100 only

TIMESTEP  Output the current PLT data for the well(s) at this time and at all subsequent timesteps.

NO    Do not output PLT data for the well(s).

DEFAULT:  NO

4. Output segment or river data?

The data is output only for multisegment wells or rivers (see "Multisegment wells" in the ECLIPSE
Technical Description). No output will be generated for other types of well

YES   Output the current segment data for the well(s) or river(s) at this time, describing flows
through each well segment.

REPT  Output the current segment data for the well(s) or river(s) at this time and at all
subsequent report times, while they are open or stopped.

ECLIPSE 100 only

TIMESTEP  Output the current segment data for the well(s) or river(s) at this time and at all
subsequent timesteps.

NO    Do not output segment data or river(s) for the well(s).

DEFAULT:  NO

End the data record with a slash (/).

End the set of records with a blank record, containing only a slash (/).

The automatic output of data at subsequent times can be cancelled by reentering WRFTPLT with the NO
option selected.

Note: In ECLIPSE 300, WRFTPLT will not output RFT or PLT data at time zero as the simulation has yet
to start.
Examples

Example 1
Request RFT data output for any currently declared well now (if open) or when it is subsequently first opened.

```
WRFTPLT
--Well     RFT    PLT   Segment
--Name     Data   Data   Data
 '*'       FOPN   /
 /   
```

Example 2
Request current PLT data output for specified wells, now and at subsequent report times.

```
WRFTPLT
--Well     RFT    PLT   Segment
--Name     Data   Data   Data
 PROD1     NO    REPT   /
 'A*'       NO    REPT   /
 /   
```

Example 3
Request reach data output for specified river, now and at subsequent report times.

```
WRFTPLT
--River     RFT    PLT   River
--Name     Data   Data   Data
 MYRIVER    NO    NO    REPT/
 /   
```
Sets salt concentrations for injection wells

This keyword is used to specify the concentration of salt in the injection stream of each well. The keyword should only be used in runs with the brine option active (keyword BRINE in the RUNSPEC section). In runs with the polymer option active (keyword POLYMER in the RUNSPEC section) the keyword WPOLYMER should be used instead of WSALT. If neither of the WSALT or WPOLYMER keywords appear in the injection schedule, then concentration values of zero are assumed. Data should only appear for wells which are currently declared to be water injection wells.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1 Well name, well name template, well list or well list template

A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2 to [N+1] The concentration of salt in the injection stream for the well.

In the standard brine model, there is a single salt component. In this case, item 2 contains a single value which is the salt concentration of the injection stream. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

ECLIPSE 100 only

When the multi-component brine model (keyword BRINE in conjunction with keyword ECLMC) is active, the BRINE keyword has N entries which specify all of the salt components in the brine mixture. In this case, items 2 to [N+1] of the WSALT keyword contain entries for each of these salt components in the injection stream. Any or all of the concentrations can be user defined arguments (UDA), and can be set with keywords UDQ and UDADIMS.

UNITS: kg/sm³ (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm³ (PVT-M)

[N+2] The group’s salt concentration to be used for the injection stream

This item specifies the group whose produced salt concentration should be used for the salt concentration of the injection stream for the well.

If no group is specified, the concentrations in items 2 to [N+1] will be used.

Note: When using group reinjection the concentration used by the well will lag the group’s concentration by one timestep.

Note: Group reinjection is not available in the Multi-Component Brine model.

End data record with a slash (/).

End the set of records with a blank record, containing just a slash (/).
Examples

Example 1

```
WSALT
  INJ1  0.8 /
  INJ2  0.7 /
  INJ3  0.7 /
/
```

Example 2

*ECLIPSE 100 only*

Multi-component brine model with three salts.

```
WSALT
  -- Well Salt1 Salt2 Salt3
  INJ1  0.8  0.2  0.01 /
  INJ2  0.7  0.3  0.02 /
/
```
WSCCLEAN Multiplies deposited well scale by a given value

This keyword can be used to reduce the amount of scale deposited around well connections after a scale cleanup operation. The rate at which scale is deposited around well connections and how the currently deposited scale alters the productivity index of the connections is controlled by tables defined in the SCDPTAB and SCDATAB keywords; these tables are assigned to wells using the WSCTAB keyword. To change the deposited scale of all the connections in a well, leave items 3 - 7 defaulted. To change the deposited scale of a subset of connections in a well, you can identify the subset by their I,J,K location (items 3 - 5). A subset of connections can also be identified by their completion number (items 6 and 7). Completion numbers are set in keyword COMPLUMP.

The WSCCLEAN keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Multiplier to act on the amount of currently deposited scale around well connections
   The value must not be negative and should be less than 1.0 for a cleanup operation
   DEFAULT: 1.0

3. I - location of connecting grid block(s)
   DEFAULT: Negative (allows any I-location)

4. J - location of connecting grid block(s)
   DEFAULT: Negative (allows any J-location)

5. K - location of connecting grid block(s)
   DEFAULT: Negative (allows any K-location)

6. Number of first completion in range
   DEFAULT: Negative; allows any completion number up to item 7 (if set)

7. Number of last completion in range
   DEFAULT: Negative; allows any completion number from item 6 (if set)

If items 3 - 7 are all defaulted, the multiplier applies to all connections in the well.

If any of items 3 - 7 are set positive, the multiplying factor applies only to a subset of the connections in the well. The connections in the subset are those with I,J,K locations matching the specified location indices in items 3 - 5 and a completion number in the range specified by items 6 - 7. A zero or negative value for a location index (I, J or K) matches any value of that location index. Similarly a zero or negative value for a completion range lower/upper end-point indicates that there is no lower/upper bound on the completion numbers.
Notes

- The total amount of scale deposited around a connection, the scale deposited per unit perforated length, and resulting PI reduction factor can each be written to the Summary file using the SUMMARY section keywords CDSM, CDSML and CDSF.

- When WSCCLEAN is read, the deposited scale of the well’s connections (or a chosen set thereof) is immediately multiplied by the factor specified in this keyword. At the beginning of the next timestep, these reduced scale depositions are used to recalculate connection PI reduction factors that apply throughout the timestep. At the end of this timestep, the reported scale deposition is the sum of the reduced scale deposition at the beginning of the timestep and the scale deposited during the timestep.

- This keyword can be used with OPTIONS3 (item 43) to avoid an error message being generated when a local grid well is referred to.

Example

```plaintext
WSCCLEAN
    PROD1  0.9 / (all connections in the well PROD1)
    PROD2  0.82 1* 1* 5 / (all connections of well PROD2 in layer 5)
    PROD3  0.6 1* 1* 1* 7 7 / (all connections of PROD3 in completion number 7)
```

Keywords

2659
WSCCLENL  Multiplies deposited well scale by a given value within local grids

This keyword can be used to reduce the amount of scale deposited around well connections after a scale cleanup operation. The rate at which scale is deposited around well connections and how the currently deposited scale alters the productivity index of the connections is controlled by tables defined in the SCDPTAB and SCDATAB keywords; these tables are assigned to wells using the WSCTAB keyword. To change the deposited scale of all the connections in a well, leave items 3 - 8 defaulted. To change the deposited scale of a subset of connections in a well, you can identify the subset by their local grid name and their I,J,K location (items 4 - 6). A subset of connections can also be identified by their completion number (items 7 and 8). Completion numbers are set in keyword COMPLMPL.

WSCCLENL performs the same role as WSCCLEAN, but the name of the local grid containing the connections specified in the record should be entered in item 3 of WSCCLENL. This keyword should be used when the well is completed in a local grid.

The WSCCLENL keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Multiplier to act on the amount of currently deposited scale around well connections

   The value must not be negative and should be less than 1.0 for a cleanup operation

   DEFAULT: 1.0

3. Name of local grid containing the connections in items 4 - 6

   If this item is defaulted or set to a blank string, the local grid name is set to the well’s local grid named in keyword WELSPECL. (In ECLIPSE 100, the local grid name should not be defaulted if the local grid is amalgamated.)

   The local grid name may also be defaulted if the connection locations in items 4, 5 and 6 are all defaulted. The multiplier in item 2 then applies to all the connections in the well (or to specific completions within it if items 7 and 8 are set).

   DEFAULT: ‘’

4. I - location of connecting grid block(s)

   DEFAULT: Negative (allows any I-location)

5. J - location of connecting grid block(s)

   DEFAULT: Negative (allows any J-location)

6. K - location of connecting grid block(s)

   DEFAULT: Negative (allows any K-location)

7. Number of first completion in range
DEFAULT: Negative; allows any completion number up to item 8 (if set)

8. Number of last completion in range

DEFAULT: Negative; allows any completion number from item 7 (if set)

If items 4 - 8 are all defaulted, the multiplier applies to all connections in the well.

If any of items 4 - 8 are set positive, the multiplying factor applies only to a subset of the connections in the well. The connections in the subset are those with I,J,K locations matching the specified location indices in items 4 - 6 in the local grid named in item 3, and a completion number in the range specified by items 7 - 8. A zero or negative value for a location index (I, J or K) matches any value of that location index. Similarly a zero or negative value for a completion range lower/upper end-point indicates that there is no lower/upper bound on the completion numbers.

Notes

• The total amount of scale deposited around a connection, the scale deposited per unit perforated length, and resulting PI reduction factor can each be written to the Summary file using the SUMMARY section keywords CDSM, CDSML and CDSF.

• When WSCCLENL is read, the deposited scale of the well’s connections (or a chosen set thereof) is immediately multiplied by the factor specified in this keyword. At the beginning of the next timestep, these reduced scale depositions are used to recalculate connection PI reduction factors that apply throughout the timestep. At the end of this timestep, the reported scale deposition is the sum of the reduced scale deposition at the beginning of the timestep and the scale deposited during the timestep.

Example

WSCCLENL
  PROD1 0.2 / (all connections in the well PROD1)
  PROD2 0.82 LGR2 / (all connections that the well PROD2 has in LGR 'LGR2')
  PROD3 0.98 4* 5 8 / (applies to completions 5,6,7 and 8 of well PROD3)
  PROD4 0.5 LGREAST 2* 3 / (all connections in layer 3 of LGR 'LGREAST')
  PROD5 0.75 LGRWEST 3 19 7 / (Only applies to connection (3,19,7) of LGR 'LGRWEST')
**Assigns scale deposition and damage tables to wells**

This keyword may be used to assign scale deposition and scale damage tables to individual wells. These tables control the rate at which scale is deposited around well connections and how the currently deposited scale alters the productivity index of the connections. They are input with the `SCDPTAB`, `SCDATA` and `SCDATABG` keywords. The rate of scale deposition depends on the fraction of sea water present in the produced water, and a water phase tracer must accordingly be nominated to represent the sea water fraction, using keyword `SCDPTAC`.

The total amount of scale deposited around a connection, the scale deposited per unit perforated length, and resulting PI reduction factor can each be written to the Summary file using the `SUMMARY` section keywords `CDSM`, `CDSML` and `CDSF`.

Scale deposition is applied at both injecting and producing completions unless item 108 in `OPTIONS` is set in ECLIPSE 100 or item 129 in `OPTIONS3` is set in ECLIPSE 300.

The `WSCTAB` keyword is followed by any number of records, each containing the following data items and each terminated with a slash (/).

The set of records must be followed by a blank record, containing only a slash (/).

1. **Well name, well name template, well list or well list template**
   - A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the *ECLIPSE Technical Description* for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword `WLIST`.

2. **Scale deposition table number**
   - An integer not exceeding `NTSCDP` (see keyword `SCDPDIMS` in the `RUNSPEC` section).
   - Scale deposition tables must be input with keyword `SCDPTAB`.
   - A value of 0, or a negative value, means that no scale deposition table will be assigned to that well and thus no further scale is deposited around its connections.
   - DEFAULT: 0

3. **Scale damage table number**
   - An integer not exceeding `NTSCDA` (see keyword `SCDPDIMS` in the `RUNSPEC` section).
   - Scale damage tables must be input with keyword `SCDATA`.
   - A value of 0, or a negative value, means that no scale damage table is assigned to that well and thus the well performance is unaltered by the currently deposited scale.
   - DEFAULT: 0

4. **ECLIPSE 300 only for PROPFLOW**: Scale damage table number for proppant flowback
   - An integer not exceeding `NTSCDG` (see keyword `SCDPDIMS` in the `RUNSPEC` section).
   - Scale damage tables must be input with keyword `SCDATABG`.
   - A value of 0, or a negative value, means that no scale damage table is assigned to that well and thus the well performance is unaltered by the currently deposited scale.
5. **ECLIPSE 100 only for karst aquifers**: Scale dissolution table number

An integer not exceeding NTSCDE (see keyword `SCDPDIMS` in the RUNSPEC section).

Scale dissolution tables must be input with keyword `SCDETAB`.

A value of 0, or a negative value, means that no scale damage table is assigned to that well and thus the well performance is unaltered by the currently deposited scale.

DEFAULT: 0

See also keyword `WPIMULT`, which may be used to multiply well PIs by a constant factor. Alternatively, keyword `WPITAB` and its associated keyword `PIMULTAB` may be used to modify a well’s PI dynamically according to the maximum water cut it has achieved. `WSCTAB` and `WPITAB` must not both be used on the same well.

**Example**

```
WSCTAB
    'P*'     1 1 3 /
    PROD2    1 2 1 /
```

### WSEGAICD

**Defines segments to represent an autonomous inflow control device**

This keyword designates specific well segments to represent an autonomous inflow control device (ICD) in a multisegment well (see "Multisegment wells" in the ECLIPSE Technical Description). This type of device is used to control the inflow profile along a horizontal well or branch by imposing an additional pressure drop between the sand face and the tubing.

The pressure drop across the device varies with the density and viscosity of the reservoir fluid flowing through the device according to the following equation:

\[
\delta P = \left( \frac{\rho_{\text{mix}}}{\rho_{\text{cal}}} \right) \cdot \left( \frac{\mu_{\text{cal}}}{\mu_{\text{mix}}} \right)^y \cdot \rho_{\text{mix}} \cdot K \cdot q^2
\]

Eq. 3.239

where

- \( x \) is the volume flow rate exponent (set in item 13)
- \( y \) is the viscosity function exponent (set in item 14)
- \( K \) is the base strength of the AICD, with dimension of inverse area squared, defined as:

\[
K = a_{\text{AICD}} \cdot q^{x-2}
\]

Eq. 3.240

where

- \( a_{\text{AICD}} \) is defined as the strength of the AICD (see item 4)

and all the other terms are as described in the WSEGSICD keyword, apart from the viscosity of the fluid mixture at local segment conditions which is given by:

\[
\mu_{\text{mix}} = a_{o} \cdot \mu_{o} + a_{w} \cdot \mu_{w} + a_{g} \cdot \mu_{g}
\]

Eq. 3.241

where

- \( a_{o,w,g} \) is the volume fraction of the free oil, water and gas phases at local conditions.
- \( \mu_{o,w,g} \) is the viscosity of the oil, water and gas phases at local conditions.

**Note:** The guidance given in the WSEGSICD keyword on the placement of such inflow control device segments should also be followed for WSEGAICD segments.

The WSEGAICD keyword is followed by a number of data records, each containing the following items of data. Unless otherwise stated, entering a negative value for an item defaults the item. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

1. **Well name, well name template, well list or well list template**
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.
The well must previously have had its segments defined with keyword \texttt{WELSEGS}.

2. Segment number at the start of the range
   An integer between 2 and the NSEGMX mnemonic in the \texttt{WSEGDIMS} keyword.

3. Segment number at the end of the range
   An integer between 2 and NSEGMX, not less than item 2.
   All segments in this range (that is, with segment numbers between these two values) are treated as ICD segments, and the following data items in this record apply to them. To enter data for a single segment, set item 2 and item 3 equal.

4. The strength of the AICD, $a_{\text{AICD}}$
   This is an empirical constant based on measurements of fluid flow through a particular type of ICD. The units depend on the value of the flow rate exponent, $x$.
   **UNITS:** bars/((kg/m$^3$)(rm$^3$/day)$^x$) (METRIC), psi/((lb/ft$^3$)(rft$^3$/day)$^x$) (FIELD), atm/((gm/cc)(rcc/hr)$^x$) (LAB), atm/((kg/m$^3$)(rm$^3$/day)$^x$) (PVT-M)

5. The length of the ICD, $l_{\text{ICD}}$
   This value is used (together with item 11) to determine a scaling factor which is used to scale the flow from the reservoir in order to establish the flow, $q_{\text{ICD}}$, through each ICD. The calculation is described in "Flow scaling".
   There are three possibilities for the calculation of the flow scaling factor:
   a. The scale factor is equal to the length of the ICD, $l_{\text{ICD}}$, divided by the length of the tubing segment that is the parent of the ICDs to account for the fact that the ICD segment may represent a number of ICDs in parallel. (item 11 = 0)
   b. The scale factor is equal to the absolute value of $l_{\text{ICD}}$. (item 11 = 1)
   c. The scale factor is equal to the length of the ICD, $l_{\text{ICD}}$, divided by the total length of the completions which supply the ICD. (item 11 = 2)
   item 11 allows you to specify explicitly which of these options is used. If item 11 is defaulted, then option a) is used whenever $l_{\text{ICD}}$ is positive, and option b) when $l_{\text{ICD}}$ is negative.
   **UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   **DEFAULT:** 12 m, 39.37 ft, 1,200 cm

6. The density of the calibration fluid, $\rho_{\text{cal}}$
   **UNITS:** kg/m$^3$ (METRIC), lb/ft$^3$ (FIELD), gm/cc (LAB), kg/m$^3$ (PVT-M)
   **DEFAULT:** 1000.25 kg/m$^3$ 62.416 lb/ft$^3$, 1.00025 gm/cc

7. The viscosity of the calibration fluid, $\mu_{\text{cal}}$
   **UNITS:** cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)
   **DEFAULT:** 1.45 cP
By setting item 88 of the OPTIONS keyword to 2 or 3, the mixture viscosity can be changed from a local volume fraction weighted average to the emulsion viscosity model described in "Emulsion viscosity". Note that the emulsion viscosity model is the default model for the WSEGSICD keyword.

ECLIPSE 300 only
By setting item 169 of the OPTIONS3 keyword to 2 or 3, the mixture viscosity can be changed from a local volume fraction weighted average to the emulsion viscosity model described in "Emulsion viscosity". Note that the emulsion viscosity model is the default model for the WSEGSICD keyword.

If this option is enabled, items 8-10 can be used to input the necessary parameters.

8. The critical value of the local water in liquid fraction used to select between a water-in-oil or oil-in-water equation for the emulsion viscosity; this is described in Emulsion viscosity.

UNITS: dimensionless
DEFAULT: 0.5

9. The width of the transition region about the critical value of the local water in liquid fraction (see item 8). For water in liquid fractions within this region, the emulsion viscosity is a linear interpolation between the water-in-oil and oil-in-water viscosity values either side of the region. This transition region is needed to ensure that viscosity is a continuous function of the water in liquid fraction as described in Emulsion viscosity.

UNITS: dimensionless
DEFAULT: 0.05

10. The maximum permitted value of the ratio of the emulsion viscosity to continuous phase viscosity. This usually only affects the water-in-oil viscosity and results in a 'plateau' region within which the emulsion viscosity is at its maximum permitted value as shown in figure 3.21. In this figure, the plateau is bounded by the transition between oil continuous and water continuous emulsions (see item 8). This is described in Emulsion viscosity.

UNITS: dimensionless
DEFAULT: 5.0

11. An integer which determines how the flow scaling factor is calculated. The calculation is described in Flow scaling.

0 The flow scale factor is equal to the ICD length (item 5) divided by the length of the tubing segment that is the parent of the ICD device, which is taken directly from the WELSEGS keyword (case a)

1 The flow scale factor is equal to the absolute value of item 5 (case b)

2 The flow scale factor is equal to the ICD length (item 5) divided by the total length of all completions which feed the ICD (case c).

If this integer is set to a negative value then, if item 5 is positive then the flow scaling factor is equal to the ICD length divided by the length of the encased tubing segment, and if item 5 is negative then the flow scaling factor is the absolute value of item 5.

DEFAULT: -1

12. The maximum absolute value of $q_{ICD}$ for which equation 3.239 holds. This value will typically be the maximum flow-rate for which the device was calibrated. Above this value the equation will be extrapolated linearly.
UNITS: m$^3$/day (METRIC), ft$^3$/day (FIELD), cc/hr (LAB), m$^3$/day (PVT-M)

DEFAULT: No limit set

13. Volume flow rate exponent. This quantity should be greater than or equal to 0.0.


15. Flag to indicate whether the device is open or shut

OPEN   ICD is open and flow is allowed through the device

SHUT   ICD is shut and flow is not allowed through the device. Crossflow, if enabled, will still be

allowed in the branch.

DEFAULT: OPEN

End the data record with a slash (/).

End the set of data records with a slash.

Notes

- A segment can only represent a single device at any one time. Any previous devices generated with the WSEGLABY, WSEGFLIM, WSEGVALV, WSEGPULL and WSEGSICD keywords will be replaced.

- Because the device pressure drop is reported via the segment’s friction pressure drop, friction effects must be included in any multisegment well for which this keyword is specified (see item 6 of WELSEGS).

Example

```
WSEGAICD
--Well Segments ICD ICD unused scale max flow visc
--Name From To strength length param factor qicd rate fnc
--
PROD1 23 23  0.00021  1*  5*  1*  10   2.3  1.5 /
PROD1 24 24  0.00021  1*  5*  1*  10   2.8  1.5 /
PROD1 25 25  0.00042  15  5*   2   10   2.3  1.5 /
/
```

WSEGDFIN  Drift flux inclination factor for multisegment wells

This keyword may be used to enter a table of inclination factors for the Drift Flux slip model used in the multisegment wells (see "The drift flux slip model" in the ECLIPSE Technical Description). ECLIPSE will use this table to determine the inclination factor \( m(\theta) \), which multiplies the drift velocity \( v_d \) if TABLE has been requested in item 2 of keyword WSEGDFMD.

The table defaults to the results of Shi, H. et al. [Ref. 97], which is consistent with the option SHI-03 in item 1 of keyword WSEGDFMD. The built-in default table is shown in the example for this keyword. ECLIPSE will use the default table if TABLE has been requested in item 2 of keyword WSEGDFMD and WSEGDFIN has not been entered.

The keyword is followed by a single table having 3 columns and up to 10 rows. The table must be terminated with a slash (/).

Column:

1. Angle of deviation from vertical.
   The values must lie between 0 and 90 degrees, and must increase monotonically down the column.
   UNITS: degrees (METRIC), degrees (FIELD), degrees (LAB), degrees (PVT-M)

2. Inclination factor for gas-liquid slip.

3. Inclination factor for oil-water slip.

The following rules will be used to extrapolate the table outside the range of angles entered in column 1:

- If the first deviation angle is > 0 degrees, the corresponding inclination factors will apply between this angle and 0 (vertical). That is, the table is extrapolated towards zero deviation with constant inclination factors.
- If the last deviation angle is < 90 degrees, the inclination factors will be linearly extrapolated to 0.0 at 90 degrees.
- If the last deviation angle = 90 degrees, both the inclination factors must be entered as 0.0.
- For deviations > 90 degrees (where the direction of positive flow is downward), the inclination factors are assumed to vary in the same way as for upward inclinations: \( m(180 - \theta) = m(\theta) \)

The table is interpolated linearly with deviation angle. Individual values in columns 2 and 3 may be defaulted by entering 1 followed by an asterisk (1 *) instead of a number, in which case the table is interpolated linearly between the adjacent non-defaulted points.

If any drift velocity multipliers are specified in keyword WSEGDFPA (items 1 and 7) or keyword WSEGFMOD (item 5), these will be applied to the drift velocity after it is multiplied by the inclination factor.

Note: The inclination factor table is not written to the Restart files. Accordingly, it must be present in any restarted data set if it has been altered from its default values with this keyword.

Example

The default table of inclination factors is as follows:

```
WSEGDFIN
=-- dev  gas-liq  oil-wat
=--angle factor  factor
```
Drift flux model definition for multisegment wells

This keyword may be used to define the Drift Flux slip model used in multisegment wells (see "The drift flux slip model" in the ECLIPSE Technical Description). The model definition governs the default set of parameter values used in the drift flux slip calculations. If you wish to give some of the parameters different values from their model defaults, you can do so by subsequently entering the WSEGDFFPA keyword. Some of these parameters can also be set separately for individual segments with the keyword WSEGFMOD.

If you wish to use WSEGDFFMD, this keyword should be entered before the segments are defined with WELSEGS, and before changing any of the parameters from their model default values with the keyword WSEGFPA.

The WSEGDFFMD keyword is followed by a data record consisting of three items, terminated with a slash (/).

1. Drift flux model definition.

   The parameter values for each choice of model are listed in the two tables below. To see how these parameters feature in the model, refer to "The drift velocity" in the ECLIPSE Technical Description.

   ORIGINAL  The original implementation of the model.

   SHI–03   The optimized parameters determined by Shi et al (2003) [Ref. 97].


   DEFAULT: ORIGINAL

2. Inclination factor calculation.

   H–K   Uses the formula of Hasan and Kabir [Ref. 48], as recommended in the original implementation of the model. \( m(\theta) = (\cos \theta)^{0.5}(1 + \sin \theta)^2 \)

   TABLE   Allows separate inclination factors for gas-liquid slip and oil-water slip to be defined as a table vs. inclination angle, entered with keyword WSEGDFFIN.

   If no table is entered with WSEGDFFIN, a built-in table will be used that reflects the results of Shi et al (2003) [Ref. 97], which is consistent with the option SHI–03 in item 1.

   SHI–04   Uses the formulae determined by Shi et al (2004) [Ref. 96]:

   \[
   m(\theta) = 1.85(\cos \theta)^{0.21}(1 + \sin \theta)^{0.95} \quad \text{for gas-liquid slip, and}
   \]

   \[
   m(\theta) = 1.07\cos \theta + 3.23 \sin 2\theta - 2.32\sin 3\theta; (\theta \leq 88^\circ) \quad \text{for oil-water slip}
   \]

   DEFAULT: H–K

3. Gas effect on oil-water slip.

   NO   Oil-water slip is unaffected by the presence of free gas.

   SHI–04   Free gas disrupts the slip between oil and water, depending on the pipe inclination, as described by Shi et al (2004) [Ref. 96].

   See "Three-phase flow and oil-water slip" in the ECLIPSE Technical Description.
DEFAULT: NO

End the data record with a slash (/).

<table>
<thead>
<tr>
<th>Parameter</th>
<th>ORIGINAL</th>
<th>SHI-03</th>
<th>SHI-04</th>
</tr>
</thead>
<tbody>
<tr>
<td>Profile coefficient A</td>
<td>1.2</td>
<td>1.2</td>
<td>1.0</td>
</tr>
<tr>
<td>Profile coefficient B</td>
<td>0.3</td>
<td>0.6</td>
<td>irrelevant</td>
</tr>
<tr>
<td>Profile multiplier $F_v$</td>
<td>1.0</td>
<td>1.0</td>
<td>irrelevant</td>
</tr>
<tr>
<td>Drift velocity ramp parameter $a_1$</td>
<td>0.2</td>
<td>0.05</td>
<td>0.06</td>
</tr>
<tr>
<td>Drift velocity ramp parameter $a_2$</td>
<td>0.4</td>
<td>0.13</td>
<td>0.21</td>
</tr>
</tbody>
</table>

*Table 3.52: Parameters for gas-liquid slip*

<table>
<thead>
<tr>
<th>Parameter</th>
<th>ORIGINAL</th>
<th>SHI-03</th>
<th>SHI-04</th>
</tr>
</thead>
<tbody>
<tr>
<td>Profile coefficient A</td>
<td>1.2</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>Profile coefficient $B_1$</td>
<td>0.4</td>
<td>irrelevant</td>
<td>irrelevant</td>
</tr>
<tr>
<td>Profile coefficient $B_2$</td>
<td>0.7</td>
<td>irrelevant</td>
<td>irrelevant</td>
</tr>
<tr>
<td>Drift velocity exponent $n'$</td>
<td>2.0</td>
<td>0.95</td>
<td>1.0</td>
</tr>
</tbody>
</table>

*Table 3.53: Parameters for oil-water slip*

**Example**

To use the optimized parameter set derived by Shi *et al.* and the corresponding values of the inclination factor, enter the following keyword before the segments are first defined with `WELSEGS`:

```
WSEGDFMD
SHI-03  TABLE / (This reproduces the model of Shi et al., 2003)
WSEGDFMD
SHI-04  SHI-04  SHI-04 / (Reproduces the model of Shi et al., 2004)
```
WSEGDFPA  Drift flux parameters for multisection wells

This keyword may be used to alter the parameters that control the Drift Flux slip model in multisection wells (see "The drift flux slip model" in the ECLIPSE Technical Description). Their default values depend on the Drift Flux model definition that is selected with keyword WSEGDFMD. You need to use WSEGDFPA only if you wish to alter one or more of the parameters from its default value. In this case you must enter WSEGDFPA only after defining the model with WSEGDFMD but before introducing any segments with WELSEGS.

The parameters entered here will apply to all segments using the Drift Flux model in all wells. However, it is possible to set some of these parameters separately for individual segments with the keyword WSEGFMOD, after the segments have been introduced with WELSEGS.

The WSEGDFPA keyword is followed by a data record consisting of the following items, terminated with a slash (/).

1. Multiplying factor for the gas-liquid drift velocity $v_d$
   The drift velocity is calculated as for a vertical pipe, then scaled automatically according to the segment’s inclination (see "The drift velocity" in the ECLIPSE Technical Description). A multiplying factor entered here applies to the scaled drift velocity.
   A value of 0.0 turns off the local slip between gas and liquid.
   DEFAULT: 1.0

2. Parameter $a_1$ in the expression for the gas-liquid drift velocity coefficient $K(\alpha_g)$
   This sets the start of the transition between the two expressions for $K(\alpha_g)$ (see "The drift velocity" in the ECLIPSE Technical Description).
   DEFAULT: depends on the model definition selected in WSEGDFMD.

3. Parameter $a_2$ in the expression for the gas-liquid drift velocity coefficient $K(\alpha_g)$
   This sets the end of the transition between the two expressions for $K(\alpha_g)$ (see "The drift velocity" in the ECLIPSE Technical Description).
   DEFAULT: depends on the model definition selected in WSEGDFMD.

4. Coefficient A for the gas-liquid profile parameter $C_0$
   This represents the value of the gas-liquid profile parameter at low values of the gas holdup fraction and flow velocity.
   $C_0 = \frac{A}{1 + (A-1)\beta^{2}}$
   Eq. 3.242
   A value of 1.0 turns off the profile slip.
   The value of A must not be less than 1.0
   DEFAULT: 1.2

5. Coefficient B for the gas-liquid profile parameter $C_0$
This represents the value of the gas holdup fraction or the flooding velocity fraction above which the profile parameter starts to decrease.

\[
\beta^* = \frac{(\beta - B)}{(1 - B)}
\]

Eq. 3.243

subject to \(0 \leq \beta^* \leq 1\)

where

\[
\beta = \max\left(\alpha_g, \frac{F_v}{\alpha_g |j|} \frac{v_s}{g_f}\right)
\]

Eq. 3.244

A and B must obey the relation \(B < (2 - A)/A\)

DEFAULT: depends on the model definition selected in WSEGDFMD

6. Multiplier \(F_v\) of the flooding velocity fraction in the profile reduction term \(\beta\) in the gas-liquid profile parameter \(C_0\)

A value of 1.0 reduces the profile parameter to 1.0 when the gas superficial velocity reaches the flooding velocity.

A value of 0.0 makes the profile parameter independent of the flow velocity.

DEFAULT: 1.0

7. Multiplying factor for the oil-water drift velocity \(v'_{d}\)

The drift velocity is calculated as for a vertical pipe, then scaled automatically according to the segment’s inclination (see "Three-phase flow and oil-water slip" in the ECLIPSE Technical Description). A multiplying factor entered here will be applied to the scaled drift velocity.

A value of 0.0 turns off the local slip between oil and water.

If a negative value is entered, the multiplier for the oil-water drift velocity is given the same value as the multiplier for the gas-liquid drift velocity set in item 1.

DEFAULT: Negative (implies the same value as item 1)

8. Coefficient A for the oil-water profile parameter \(C_0'\)

This represents the value of the oil-water profile parameter at low values of the oil holdup fraction. The equations for \(C_0'\) are discussed in "Three-phase flow and oil-water slip" in the ECLIPSE Technical Description.

\[
C_0' = A \text{ when } (\alpha_{ol} \leq B_1)
\]

\[
C_0' = 1 \text{ when } (\alpha_{ol} \geq B_2)
\]

\[
C_0' = A - (A - 1) \left(\frac{\alpha_{ol} B_1}{B_2 - B_1}\right) \text{ when } (B_1 < \alpha_{ol} > B_2)
\]

Eq. 3.245

DEFAULT: depends on the model definition selected in WSEGDFMD

9. Coefficient B1 for the oil-water profile parameter \(C_0'\)

DEFAULT: depends on the model definition selected in WSEGDFMD

10. Coefficient B2 for the oil-water profile parameter \(C_0'\)
In the original model, B1 and B2 must obey the relation $B1 \leq (2-A)B2$

**DEFAULT:** depends on the model definition selected in **WSEGDFMD**

11. Exponent n’ in the expression for the oil-water drift velocity $v'_d$

$$v'_d = 1.53v'_c(1-o_{ol})^{n'}$$  \hspace{1cm} \text{Eq. 3.246}

**DEFAULT:** depends on the model definition selected in **WSEGDFMD**

End the data record with a slash (/).

**Example**

```
WSEGDFPA
  1* 0.1 0.2 4* 1.0 2* 1.0 /
```
WSEGDIMS  Sets dimensions for multisegment wells

This keyword is required if the multisegment well model is used (see "Multisegment wells" in the ECLIPSE Technical Description). It reserves storage space for the multisegment well data.

The data consists of four items and must be terminated by a slash (/).

1. NSWLMX The maximum number of multisegment wells in the model.
   DEFAULT: 0

2. NSEGMX The maximum number of segments per well.
   In ECLIPSE 300, it is recommended that there should be no more than 25 segments per well, in order to ensure a reasonable level of performance if several multisegment wells are present in compositional runs.
   DEFAULT: 1

3. NLBRMX The maximum number of branches (including the main stem) per multisegment well.
   DEFAULT: 1

4. NCRDMX The maximum number of segment chord links per well.
   DEFAULT: 0

ECLIPSE 100

Large values for item 1 and, especially, item 2 will lead to poor performance in parallel. Therefore, it is recommended that you adjust these items to the actual maximum numbers.

Example

```
WSEGDIMS
  2  30  3  1 /
```
This keyword provides the capability to import or remove fluids to or from a segment in a multisegment well (see “Multisegment wells” in the ECLIPSE Technical Description), either at a specified rate or as a function of the segment’s pressure. The source or sink for this fluid is external to the reservoir grid. The available options are:

- Import water or gas at a rate specified in item 4.
- Import water or gas at a rate given by
  \[ Q = R (P_{\text{ext}} - P_{\text{seg}}) \]  
  while \( P_{\text{ext}} > P_{\text{seg}} \), where
  
  - \( Q \) is the volumetric import rate measured at surface conditions
  - \( R \) is a constant set in item 5
  - \( P_{\text{ext}} \) is an external pressure set in item 6
  - \( P_{\text{seg}} \) is the segment’s pressure
  
  The import rate is subject to an upper limit set in item 4.

- Remove segment fluid at a rate given by
  \[ Q = R (P_{\text{seg}} - P_{\text{ext}}) \]  
  while \( P_{\text{seg}} > P_{\text{ext}} \), where
  
  - \( Q \) is the volumetric rate of fluid removal
  - \( R \) and the external pressure \( P_{\text{ext}} \) are set in items 5 and 6.
  
  The fluid removal rate can either be expressed as an equivalent wet gas volume rate or an equivalent water volume rate, depending on the entry (FREMP or WREMP) in item 3. The choice should depend on whether the fluid in the segment is predominantly hydrocarbon or water/steam. The wet gas rate is related to the moles removed using a unity Z-factor at surface conditions. The equivalent water volume rate is related to the moles removed using the density of water at surface conditions.
  
  The removal rate is subject to an upper limit set in item 4.

The composition of the removed fluid is the current composition in the segment.

There are a number of possible applications for this facility. Gas import may be used to model the effects of lift gas injection. Water import may be used to take account of the water used to power a downhole hydraulic pump if this is injected into the wellbore. The fluid removal facility is intended for use with the Thermal Option to model recirculating wells.

When a well has an external source/sink, the reported production (or injection) rate refers to the fluid flowing out of (or into) the top of the well. Thus, for a production well with fluid import from an external source, the reported production rate will be the fluid production from the formation plus the import rate. For an injection well with fluid import, the fluid injected into the formation is the sum of the reported injection rate and the fluid import rate.

Fluid import or removal is turned off whenever the well is shut.
If injection or production from a well is entirely determined by import or removal, it is better to specify the well as stopped, for example with a \texttt{WCONINJE} or \texttt{WCONPROD} keyword, than to specify the well as an injector or producer with a zero rate.

When fluid import is requested, a smaller rate/unit drawdown usually produces better convergence and robustness.

Wells that have external source/sinks usually run more robustly when they are allowed to crossflow.

Segment fluid import/removal rates and cumulative totals may be written to the Summary file using the \texttt{SUMMARY} section mnemonics \texttt{S(W,G)IM(R,T)} in ECLIPSE 100 and \texttt{S(O,W,G)(IM,RM)(R,T)} in ECLIPSE 300.

\textit{ECLIPSE 100 only} Fluid imported will be pure water or gas; it will not contain tracers or other components such as polymer, brine, surfactant, foam or solvent.

The \texttt{WSEGEXSS} keyword is followed by a number of data records to define the well, the location of the segment and the import/removal parameters. Only one external source/sink is allowed per segment. If a segment is specified more than once, its previous source/sink parameters are replaced with the new data.

Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

The data items in each record are:

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the \textit{ECLIPSE Technical Description} for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword \texttt{WLIST}.
   The well must previously have had its segments defined with keyword \texttt{WELSEGS}.

2. Segment number containing the external source/sink
   An integer between 2 and the \texttt{NSEGMX} mnemonic in the \texttt{WSEGDIMS} keyword.

3. A character string specifying the source/sink type
   The string must be one of the following:
   \begin{itemize}
   \item \texttt{WIMPR} Water import at a specified rate
   \item \texttt{GIMPR} Gas import at a specified rate
   \item \texttt{WIMPP} Water import with a specified external pressure, equation 3.247
   \item \texttt{GIMPP} Gas import with a specified external pressure, equation 3.247
   \end{itemize}

\textit{ECLIPSE 300 only} Fluid removal with a specified external pressure, equation 3.247, with the removal rate expressed as an equivalent \texttt{wet gas} volume rate

\textit{ECLIPSE 300 only} Fluid removal with a specified external pressure, equation 3.247, with the removal rates expressed as an equivalent \texttt{water} volume rate

\texttt{NONE} This removes the external source/sink from the segment
4. Volumetric import rate of water or gas at surface conditions, for WIMPR or GIMPR, or upper limit to the water or gas import rate, for WIMPP or GIMPP or the upper limit to the water or wet gas removal rate, for WREMP or FREMP.

   **UNITS:** sm³/day (METRIC), stb/day or Mscf/day (FIELD), scc/hr (LAB), sm³/day (PVT-M)

   **DEFAULT:** No upper limit, for WIMPP, GIMPP, WREMP or FREMP; 0.0 for WIMPR or GIMPR

5. \( R \), the import/removal rate per unit pressure difference, for WIMPP, GIMPP, FREMP or WREMP

   **UNITS:** sm³/day/bars (METRIC), stb/day/psi or Mscf/day/psi (FIELD), scc/hr/atm (LAB), sm³/day/atm (PVT-M)

   **DEFAULT:** 0.0

6. \( P_{ext} \), the external pressure, for WIMPP, GIMPP, FREMP or WREMP

   **UNITS:** barsa (METRIC), psia (FIELD), atm (LAB), atm (PVT-M)

   **DEFAULT:** 0.0

   **ECLIPSE 300 only** This item must also be set if WIMPR is selected when running the Thermal Option. The pressure is used together with the temperature (specified in item 8) to calculate the enthalpy of the imported water.

   **ECLIPSE 300 only** 7. Wellstream name to specify the imported hydrocarbon composition, for GIMPR or GIMPP

   The wellstream composition must be specified with keyword WELLSTRE.

   If this item is defaulted, the imported gas composition is set to that of the field separator gas.

   **ECLIPSE 300 only** 8. Temperature of imported fluid

   **UNITS:** °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

   This item must be set if WIMPR, GIMPR, WIMPP or GIMPP is selected when running the Thermal Option. Otherwise, this item should be left defaulted.

   **Thermal option only** 9. Quality of imported water

   If the source/sink type is WIMPR or WIMPP in item 3 above, then the quality of the imported water can be set. In this case, either the external pressure in item 6 must be set, or the external temperature in item 8 must be set. If both are set, then the external pressure will be used.

   **UNITS:** dimensionless

End the data record with a slash (/).

End the set of data records with a slash.

**Example**

An ECLIPSE 300 example:

| WSEGEXSS | Name Seg Type Rate R Pext Stream Temp |
|----------|------------------------------------|-----------------|
| I        | 4  FREMP  5000  100  2000 /     |
| P        | 7  WIMPR  800 /                  |
TSTEP

-- Remove external source from segment 7 of well P
WSEGEXSS
  P   7   /
WSEGFLIM  Defines segments to represent a flow limiting valve

This keyword designates specific well segments to represent a flow limiting valve in a multisegment well (see "Multisegment wells" in the ECLIPSE Technical Description). This is a hypothetical device that limits the flow rate of oil, water or gas (at surface conditions) through the segment to a specified maximum value, by sharply increasing the frictional pressure drop across the segment if the limit is exceeded. The device can be used to

- limit the production of associated water and gas through a section of tubing
- control the distribution of production or injection between the branches of a multilateral well
- prevent branch-to-branch crossflow in a multilateral well.

The frictional pressure drop across the segment is calculated as

\[ \delta P_f = \delta P_1 + \delta P_2 \]

Eq. 3.249

where

\[ \delta P_1 = A_1(Q_1 - Lim_1) \quad \text{when} \quad (Q_1 > Lim_1) \]

\[ \delta P_2 = A_2(Q_2 - Lim_2) \quad \text{when} \quad (Q_2 > Lim_2) \]

Eq. 3.250

The \( A \) terms are coefficients, which should be large to provide a steeply increasing pressure drop. The \( Q \) terms represent the flow rate of a chosen phase (oil, water, gas or liquid) through the segment, at surface conditions, while the \( Lim \) terms represent their specified limiting values. It is therefore possible to limit two flows in one valve segment. Each pressure drop term is zero while its flow limit is not exceeded. The hydrostatic and acceleration pressure drop components are calculated in the usual way and added to the friction pressure drop. The flow is treated as homogeneous within these segments; there is no slip between the phases.

If a \( Lim \) term is set negative, its pressure drop term acts to impede the negative flow when its magnitude increases beyond the limit; it does not prevent positive flows. The term becomes

\[ \delta P = A(Q - Lim) \quad \text{when} \quad (Q < Lim < 0) \]

Eq. 3.251

Thus negative limits can be used to limit flows in injection well branches, and small negative flow limits can be used to restrict crossflow in production wells.

Because of the non-linear nature of the pressure drop term, it is advisable to use the keyword WSEGITER to activate the more robust iteration scheme.

Care should be taken to ensure that a flow limiting valve does not conflict with the overall flow control of the well; this would be the case if it was placed downstream of all connections in a producer or upstream of all connections in an injector and the limited phase was the same as the control phase for the well.

See also keyword WSEGMULT which provides an alternative means of impeding flow at high values of the WOR or GOR, and the section on “The pressure loss calculation” in the ECLIPSE Technical Description.

The WSEGFLIM keyword is followed by a number of data records, each containing the following items of data. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names
should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword **WLIST**.

The well must previously have had its segments defined with keyword **WELSEGS**.

2. **Segment number at the start of the range**
   An integer between 2 and the **NSEGMX** mnemonic in the **WSEGDIMS** keyword.

3. **Segment number at the end of the range**
   An integer between 2 and **NSEGMX**, not less than item 2.

   All segments in this range (that is, with segment numbers between these two values) will be treated as flow limiting valve segments, and the following data items in this record are applied to them. To enter data for a single segment, set items 2 and 3 equal.

4. **Phase limited in the first pressure drop term**
   - **OIL**: The total oil flow rate at surface conditions
   - **WAT**: The water flow rate at surface conditions
   - **GAS**: The total gas flow rate at surface conditions
   - **LIQ**: The liquid flow rate (oil + water) at surface conditions
   - **NONE**: The liquid flow rate (oil + water) at surface conditions

   **DEFAULT**: NONE

5. **The flow limit in the first pressure drop term**
   The limit applies to the phase defined in item 4. A positive value restricts production flows while a negative value restricts injection flows.

   **UNITS**: sm$^3$/day (**METRIC**), stb/day or Mscf/day (**FIELD**), scc/hr (**LAB**), sm$^3$/day (**PVT-M**)

6. **The coefficient A in the first pressure drop term**
   The larger the value, the more effective the operation of the valve is in preventing the flow rate exceeding the limit. If the flow limit is being exceeded by a significant amount, the value of this coefficient should be increased. However, too high a value may retard the convergence of the well iterations.

   The coefficient must not be negative.

7. **Phase limited in the second pressure drop term**
   The choices are the same as for item 4.

   **DEFAULT**: NONE

8. **The flow limit in the second pressure drop term**
   The limit applies to the phase defined in item 7. A positive value restricts production flows while a negative value restricts injection flows.

   **UNITS**: sm$^3$/day (**METRIC**), stb/day or Mscf/day (**FIELD**), scc/hr (**LAB**), sm$^3$/day (**PVT-M**)

9. **The coefficient A in the second pressure drop term**
10. Flag to indicate whether the device is open or shut
   
   **OPEN**  ICD is open and flow is allowed through the device
   
   **SHUT**  ICD is shut and flow is not allowed through the device. Crossflow, if enabled, will still be allowed in the branch.

   **DEFAULT:** **OPEN**

   End the data record with a slash (/).

   End the set of data records with a slash.

   If both items 4 and 7 are **NONE**, the segment ceases to act as a flow limiting valve and its pressure drop is calculated as for a normal segment.

   **Note:** A segment can only represent a single device at any one time. Any previous devices generated with the **WSEGLABY, WSEGVALV, WSEGPULL, WSEGSICD** and **WSEGAICD** keywords will be replaced.

**Example**

In the top branch of well **PROD1** (segment 12) limit the water production to 1000 stb/day.

In the bottom branch of well **PROD1** (segment 20) prevent crossflow of oil and limit the gas production to 2500 Mscf/day.

Stop the crossflow of water (that is water production) in segment 10 of the water injection well **WINJ1**.

```
WSEGFLIM
-- Well    Segments   Phase 1   Lim 1   A 1   Phase 2   Lim 2   A 2
-- Name    From  To
PROD1    12   12     WAT     1000    100  /
PROD1    20   20     OIL     -1      100     GAS     2500    100/
WINJ1    10   10     WAT      1      100  /
/  
/  
```
This keyword specifies the multiphase flow model that is used to calculate the pressure losses within individual segments of a multisegment well (see "Multisegment wells" in the ECLIPSE Technical Description). The choices are a no-slip homogeneous flow model, in which all the phases flow with the same velocity, and a drift flux model, which allows slip between the phases and can result in countercurrent flow - heavy and light phases flowing in opposite directions - at low flow rates (see "The drift flux slip model" in the ECLIPSE Technical Description). Alternatively, individual segments can have their pressure losses either obtained from a VFP table (see keyword WSEGTABL) or calculated from a built-in model of a particular flow control device (see keywords WSEGVALV, WSEGFLIM and WSEGLABY).

Segments not specified in keyword WSEGFMOD, WSEGTABL, WSEGVALV, WSEGFLIM or WSEGLABY will use the default flow model selected in item 7 of Record 1 of keyword WELSEGS.

The WSEGFMOD keyword is followed by a number of data records, each containing the following items of data. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.
   The well must previously have had its segments defined with keyword WELSEGS.

2. Segment number at the start of the range
   An integer between 2 and NSEGMX (see item 2 in the keyword WSEGDIMS keyword).

3. Segment number at the end of the range
   An integer between 2 and NSEGMX, not less than item 2.
   The following data items in this record are applied to all the segments in the range. To enter data for a single segment, set items 2 and 3 equal.

4. Multiphase flow model used for pressure drop calculations
   HO  No-slip homogeneous flow model.
   All the phases flow with the same velocity.
   DF  Drift flux model.
   Slip between the phases is a combination of profile slip (a velocity ratio) and local slip (a vertical relative velocity). At low flow rates the local slip can cause countercurrent flow, when the heavy and light phases flow in opposite directions.

   If the drift flux model is requested for any segment, the well’s flow model in item 7 of Record 1 of keyword WELSEGS must also be set to DF.
   DEFAULT:  HO

5. Multiplying factor for the gas-liquid drift velocity (drift flux model only)
The drift velocity is calculated as for a vertical pipe, then scaled automatically according to the segment’s inclination (see "The drift velocity" in the ECLIPSE Technical Description). A multiplying factor entered here will be applied to the scaled local slip velocity.

A value of 0.0 turns off the local slip.

The oil-water drift velocity multiplier is set for all segments in item 7 of keyword \texttt{WSEGDFPA}.

**DEFAULT:** as set in item 1 of keyword \texttt{WSEGDFPA}.

6. Coefficient \(A\) for the gas-liquid profile parameter (drift flux model only)

This represents the value of the profile parameter at low values of the gas holdup fraction and flow velocity.

\[
C_0 = \frac{A}{1 + (A-1)\beta^2} \quad \text{Eq. 3.252}
\]

The equations for \(C_0\) are discussed in "The profile parameter" in the ECLIPSE Technical Description.

A value of 1.0 turns off the profile slip.

The value of \(A\) must not be less than 1.0.

The oil-water profile parameter coefficient \(A\) is set for all segments in item 8 of keyword \texttt{WSEGDFPA}.

**DEFAULT:** as specified in keyword \texttt{WSEGDFMD} or item 4 of keyword \texttt{WSEGDFPA}.

7. Coefficient \(B\) for the gas-liquid profile parameter (drift flux model only)

This represents the value of the gas holdup fraction or the flooding velocity fraction above which the profile parameter starts to decrease.

\[
\beta^* = \frac{(\beta-B)}{(1-B)} \quad \text{subject to} \quad 0 \leq \beta^* \leq 1 \quad \text{Eq. 3.253}
\]

where

\[
\beta = \max\left(\alpha_g, \frac{\alpha_g |j|}{v_{sgf}}\right) \quad \text{Eq. 3.254}
\]

A and \(B\) must obey the relation \(B < (2-A)/A\)

**DEFAULT:** as specified in keyword \texttt{WSEGDFMD} or item 5 of keyword \texttt{WSEGDFPA}.

8. Multiplier \(F_v\) of the flooding velocity fraction in the profile reduction term \(\beta\) in the gas-liquid profile parameter (drift flux model only)

A value of 1.0 reduces the profile parameter to 1.0 when the gas superficial velocity reaches the flooding velocity.

A value of 0.0 makes the profile parameter independent of the flow velocity.

**DEFAULT:** as set in item 6 of keyword \texttt{WSEGDFPA}.

9. Coefficient \(B1\) for the oil-water profile parameter \(C_0'\) (drift flux model only)
\[ C_0' = A \quad \text{when} \quad (\alpha_{ol} \leq B1) \]
\[ C_0' = 1 \quad \text{when} \quad (\alpha_{ol} \geq B2) \]
\[ C_0' = A - (A-1) \left( \frac{\alpha_{ol} - B1}{B2 - B1} \right) \quad \text{when} \quad (B1 < \alpha_{ol} > B2) \]

Eq. 3.255

The equations for \( C_0' \) are discussed in "Three-phase flow and oil-water slip" in the ECLIPSE Technical Description. The value of \( A \) is set from item 6.

DEFAULT: as specified in keyword WSEGDFMD or item 9 of keyword WSEGDFPA

10. Coefficient \( B2 \) for the oil-water profile parameter (drift flux model only)

\( B1 \) and \( B2 \) must obey the relation \( B1 \leq (2-A)B2 \)

DEFAULT: as specified in keyword WSEGDFMD or item 10 of keyword WSEGDFPA

End the data record with a slash (/).

End the set of data records with a slash.

**Note:** If WSEGFMOD is applied to a segment that has previously been declared as a sub-critical valve (WSEGVALV), a flow limiting valve (WSEGFLIM), a labyrinth device (WSEGLABY) or a VFP table lookup segment (WSEGTABL), it loses its former attributes and become a standard segment using either the homogeneous or the drift flux flow model according to item 4.

The top segment (segment 1) always uses the homogeneous flow model. There are no pressure losses calculated along this segment.

**Example**

<table>
<thead>
<tr>
<th>WSEGFMOD</th>
</tr>
</thead>
<tbody>
<tr>
<td>-- Well Segments Flow Local Prof Prof Prof Prof Prof Prof</td>
</tr>
<tr>
<td>Name From To Model Slip A B v mult B1 B2</td>
</tr>
<tr>
<td>PROD1 2 6 DF 1.1 1.3 0.3 1.0 0.5 0.9 /</td>
</tr>
<tr>
<td>PROD1 7 12 DF /</td>
</tr>
<tr>
<td>/</td>
</tr>
</tbody>
</table>
Segment heat transfer coefficient

In a Thermal run, this keyword allows a heat transfer coefficient to be specified at a segment in a multisegment well (see "Multisegment wells" in the ECLIPSE Technical Description). Heat transfer can take place from the segment to the reservoir grid, to another segment, or to a specified fixed external temperature. The well’s segment structure must previously have been defined with the WELSEGS keyword.

The heat transfer rate $Q_{ht}$ to/from the segment is

$$Q_{ht} = L \cdot \frac{(T_{seg} - T_{is,ic,ex})}{R_h}$$

where

$L$ is the thermal contact length. See "Thermal contact lengths" in the ECLIPSE Technical Description

$T_{seg}$ is the temperature in the segment,

$T_{is,ic,ex}$ is the temperature of a target segment or completion grid block or external fixed temperature

$R_h$ is the specific thermal resistance.

See "Conductive heat transfer" in the ECLIPSE Technical Description

For heat transfer between segments, a semi-implicit heat transfer rate is used, given by

$$Q_{ht}^{N+1} = S_{int} \cdot Q_{ht} + (1 - S_{int}) \cdot Q_{ht}^N$$

where

$S_{int}$ is an interpolation constant and

$Q_{ht}^N$ is the $N$ time level heat transfer.

The keyword is followed by a number of data records to define the well, a segment range, type of heat transfer coefficient and the target segment or external temperature. If a segment is specified more than once, then its properties are changed. This option is not active when wells are shut.

Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

The data items in each record are:

1. Well name
2. Segment number at the start of the range
   
   An integer between 1 and NSEGMX (see item 2 in the WSEGDIMS keyword).
3. Segment number at the end of the range
   
   An integer between 1 and NSEGMX.
4. A character string specifying the heat transfer coefficient type:
   
   The string must be one of the following:
COMP  Heat transfer to a previously defined completion. (This replaces all previously defined heat transfer coefficients.)

SEG   Heat transfer to a another segment. (This replaces all previously defined heat transfer coefficients.)

TEMP  Heat transfer to a specified fixed external temperature. (This replaces all previously defined heat transfer coefficients.)

NONE  This is equivalent to the default type and removes the segment from the list of external source/sinks. (This deletes all previously defined heat transfer coefficients.)

The above definitions will replace or delete all previously defined heat transfer coefficients for this segment. If more than one heat transfer coefficient is required, a maximum of one completion transfer coefficient, one formation transfer coefficient and up to five segment-to-segment heat transfer coefficients may be specified concurrently by appending + to COMP, SEG or TEMP above. These will be interpreted as follows:

COMP+  Add a heat transfer to a previously defined completion to the existing set of heat transfer coefficients. This will add a previously undefined or replace a previously defined transfer coefficient of this type but will not alter other heat transfer coefficients currently associated with this segment.

SEG+   Add a heat transfer coefficient to another segment. If this has not been previously specified, this will add a new segment-to-segment heat transfer coefficient to the set associated with this segment up to a maximum of five. If this has been previously specified, the properties will be updated. If the addition of a new transfer coefficient would cause the maximum to be exceeded, the new transfer coefficient will be ignored.

TEMP+  Add a heat transfer to a specified fixed external temperature to the existing set of heat transfer coefficients. This will add a previously undefined or replace a previously defined transfer coefficient of this type but will not alter other heat transfer coefficients currently associated with this segment.

One or more previously defined heat transfer coefficients may be also removed without removing the remainder by appending – to COMP, SEG or TEMP above. These will be interpreted as follows:

COMP−  Remove a heat transfer coefficient to a previously defined completion from the existing set of heat transfer coefficients. This will not alter other heat transfer coefficients currently associated with this segment.

SEG−   Remove a heat transfer coefficient to a another segment from the existing set of heat transfer coefficients. This will not alter other heat transfer coefficients currently associated with this segment.

TEMP−  Remove a heat transfer to a specified fixed external temperature from the existing set of heat transfer coefficients. This will not alter other heat transfer coefficients currently associated with this segment.

5. Specific Thermal Resistance:

UNITS: (kJ/m/day/K)$^{-1}$ (METRIC), (Btu/ft/day/$^\circ$R)$^{-1}$ (FIELD), (J/cm/hr/K)$^{-1}$ (LAB), (kJ/m/day/K)$^{-1}$ (PVT–M)
6. Target segment number if item 4, the heat transfer coefficient type, is SEG:
   An integer between 1 and NSEGMX mnemonic in the WSEGDIMS keyword.
7. External fixed temperature if item 4, the heat transfer coefficient type, is TEMP:
   UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
8. Interpolation constant if item 4, the heat transfer coefficient type, is SEG:
   DEFAULT: 0.5
9. Contact length, if item 4, the heat transfer coefficient type, is SEG or TEMP
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   DEFAULT: \((L_n + L_{n+1})/2\) where \(L_n\) and \(L_{n+1}\) are the lengths of segments either side of the segment node.

**Example**

```
-- Define heat transfer coefficients for segments 2 to 4 and segment 5
WSEGHEAT
-- Name IS1 IS2  Type  Resist  IST
 I  2  4  COMP  0.02 /
 I  5  5  SEG  0.04  24 /
 /
TSTEP
...
/
-- Remove all heat transfer coefficients associated with segment 2
WSEGHEAT
-- Name IS1 IS2  Type
 I  2  2  NONE /
 /
-- Add another heat transfer coefficient to segment 5
WSEGHEAT
-- Name IS1 IS2  Type  Resist  IST
 I  5  5  SEG+  0.02  25 /
 /
```
WSEGINIT

Sets initial conditions for well segments

This keyword may be used to specify the initial conditions for segments in a multisegment well (see "Multisegment wells" in the ECLIPSE Technical Description), namely the pressure and the initial fluid contents in each segment.

It will normally be adequate to let ECLIPSE set the initial conditions of new wells automatically. However, it could be useful to set them manually if you are interested in modeling the very early time behavior of the well immediately on startup. Segments that are not initialized with this keyword have their initial conditions set automatically, according to the following rules. For production wells, the initial fluid contents approximately reflect the average saturations in the connecting grid blocks, and the pressure is set somewhat below that of the connecting grid blocks. For injection wells, the segments contain the injected fluid and the pressure is set somewhat above that of the connecting grid blocks.

The WSEGINIT keyword is followed by a number of data records, each containing the following items of data. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.
   The well must previously have had its segments defined with keyword WELSEGS.

2. Segment number at the start of the range
   An integer between 1 and NSEGMX, which is the maximum number of segments per well (item 2 in WSEGDIMS).

3. Segment number at the end of the range
   An integer between 1 and NSEGMX, not less than item 2 above.
   The following initial conditions in this record apply to all the segments in the range. To enter the initial conditions for a single segment, set items 2 and 3 equal.

4. Initial pressure
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

5. Initial water holdup fraction
   This is the fraction of the volume of the segment that is occupied by water. The value must lie between 0.0 and 1.0.

6. Initial gas holdup fraction
   This is the fraction of the volume of the segment that is occupied by free gas. The value must lie between 0.0 and 1.0.

7. Dissolved gas concentration in the oil, Rs, in runs with live oil
   ECLIPSE limits this so as not to exceed the saturated value at the initial pressure.
   UNITS: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)
   DEFAULT: 0.0
8. Vaporized oil concentration in the gas, Rv, in runs with wet gas

   ECLIPSE limits this so as not to exceed the saturated value at the initial pressure.

   UNITS: \( \text{sm}^3/\text{sm}^3 \) (METRIC), \( \text{stb/Mscf} \) (FIELD), \( \text{scc/scc} \) (LAB), \( \text{sm}^3/\text{sm}^3 \) (PVT-M)

   DEFAULT: 0.0

9. API gravity of the oil

   This item must be set if the API tracking option is active (keyword API in the RUNSPEC section). It is ignored in runs that do not use this option.

   UNITS: °API

10. Polymer concentration

    This item may be used if the polymer flood model is active (keyword POLYMER in the RUNSPEC section).

    UNITS: \( \text{kg/sm}^3 \) (METRIC), \( \text{lb/stb} \) (FIELD), \( \text{gm/scc} \) (LAB), \( \text{kg/sm}^3 \) (PVT-M)

    DEFAULT: 0.0

11. Brine concentration

    This item may be used if the polymer flood model is active (keyword POLYMER in the RUNSPEC section).

    UNITS: \( \text{kg/sm}^3 \) (METRIC), \( \text{lb/stb} \) (FIELD), \( \text{gm/scc} \) (LAB), \( \text{kg/sm}^3 \) (PVT-M)

    DEFAULT: 0.0

End the data record with a slash (/).

End the set of data records with a slash.

**Example**

```
WSEGINIT
-- Name    Segments   Pres    Hw    Hg    Rs    Rv    API
PROD1     1  10    3500    0.0   1.0 / gas region
PROD1    11  20    3500    0.0   0.0   0.9 / oil region
PROD1    21  30    3500    1.0   0.0 /water region
/ 2690
```
WSEGITER  Iteration parameters for multisegment wells

This keyword controls the iteration sequence for the solution of multisegment wells (see "Multisegment wells" in the ECLIPSE Technical Description). These wells are often harder to converge than 'standard' wells, and would benefit from a more robust iteration sequence. By default, if the WSEGITER keyword is not entered, each well is allowed up to MXWSIT iterations (set in item 5 of the 3rd record of keyword TUNING) to converge its solution. If it cannot converge in this number of iterations a ‘problem’ message is issued and the latest (unconverged) well solution is used. This may hamper the convergence of the non-linear iterations of the timestep. However, if the WSEGITER keyword is entered ECLIPSE invokes a sequence of iteration cycles that gives the well a greater chance of converging its solution. The method, which applies only to multisegment wells, is described below.

In ECLIPSE 300, the WSEGITER convergence sequence is only called after a predictor is called. If a well does not converge in a cycle of MXWSIT iterations, the pressure equation in each of its segments is altered by averaging its pressure drop with the value at the beginning of the iteration cycle, using an adjustable weighting factor $W_p$

$$P_{\text{seg}} = P_{\text{out}} + W_p \Delta P + (1-W_p)\Delta P_{\text{prev}}$$  \hspace{1cm} \text{Eq. 3.256}

$P_{\text{seg}}$ is the pressure in a segment

$P_{\text{out}}$ is the pressure in its outlet segment (towards the wellhead)

$\Delta P$ is the calculated pressure drop across the segment

$\Delta P_{\text{prev}}$ is the pressure drop at the previous converged iteration cycle.

The effect of the weighting factor is to reduce the amount by which the segment pressure drop changes from its previous converged value. If the well does not converge in a cycle of MXWSIT iterations, the weighting factor $W_p$ is reduced by a factor $F_R$ ($W_p \leftarrow F_R \times W_p$) and a new iteration cycle is begun. If the iterations still do not converge, $W_p$ is reduced again by the same factor and another iteration cycle is begun. $W_p$ can be reduced a specified maximum number of times $N_R$ before ECLIPSE gives up and finally declares the well solution unconverged. When an iteration cycle converges with a reduced value of $W_p$ its value is increased by a factor $F_I$ ($W_p \leftarrow F_I \times W_p$) and a new iteration cycle is begun. $W_p$ is successively increased in this manner until it reaches the value 1.0, and when this iteration cycle converges the well solution is finally declared converged. Overall, this process traverses ‘difficult’ regions of the residual function by taking small steps in the pressure variable.

You must take care when selecting the iteration parameters. If the factors $F_R$ and $F_I$ are too small it can take many iteration cycles for $W_p$ to reach unity again. It can take up to $N_C$ cycles of iterations to converge a well after its maximum allowed number of reductions in $W_p$, where

$$N_c = N_R + 1 + N_R \frac{\log(1/F_R)}{\log(F_I)}$$ \hspace{1cm} \text{Eq. 3.257}

The WSEGITER keyword is followed by a data record consisting of the following items, terminated with a slash (/).

1. Maximum number of well iterations per cycle, MXWSIT
This item overwrites any value previously entered in item 5 of the 3rd record of keyword TUNING (or TUNINGL or TUNINGS).

**ECLIPSE 100**
DEFAULT: 20

**ECLIPSE 300**
DEFAULT: 40

2. Maximum number of times, \( N_R \), that a new iteration cycle with a reduced \( W_p \) will be started following an unconverged iteration cycle.

The well will finally be declared unconverged if there are more than \( N_R \) unconverged iteration cycles.

DEFAULT: 5

3. Reduction factor \( F_R \)

\( W_p \) will be reduced by this factor following an unconverged iteration cycle.

A value between 0.01 and 0.9 is allowed.

DEFAULT: 0.3

4. Increase factor \( F_I \)

\( W_p \) will be increased by this factor following each converged iteration cycle, until it reaches the value 1.0.

The value must not be less than 1.2

DEFAULT: 2.0

End the data record with a slash (/).

---

**CAUTION:** If the **TUNING** keyword is entered after **WSEGITER**, the value of **MXWSIT** in the third record of **TUNING** overwrites the value entered in item 1 of this keyword.

---

**Example**

```
WSEGITER
  25  5  0.1  3.0 /
```
WSEGLABY  Defines segments to represent a labyrinth inflow control device

This keyword designates specific well segments to represent a labyrinth inflow control device in a multisegment well (see "Multisegment wells" in the ECLIPSE Technical Description). This type of device is used to control the inflow profile along a horizontal well or branch by imposing an additional pressure drop between the sand face and the tubing. The device is placed around a section of the tubing and diverts the fluid inflowing from the adjacent part of the formation into a series of small channels before it enters the tubing. The additional pressure drop that it imposes depends upon the length of the flow path through the system of channels, which is adjustable. A series of labyrinth devices with different channel settings can be placed along the length of a horizontal well or branch, with the aim, for example, of reducing the variation in drawdown along it.

The pressure drop across the device is calculated as

\[ \delta P = \delta P_{\text{form}} + \delta P_{\text{fric}} \]  

\( \delta P_{\text{form}} \) accounts for geometric effects and the loss through the inlet section of the channel (including a pre-packed screen) and the outlet section of the channel (including a kill filter). It is calculated as

\[ \delta P_{\text{form}} = C_u (\gamma_{\text{inlet}} + (N_c - 1)\gamma_{\text{labyr}} + \gamma_{\text{outlet}}) \rho v^2 \]  

where

- \( \gamma \) is a dimensionless coefficient for each loss mechanism
- \( N_c \) is the ‘configuration number’ of the device, which determines the length of the channel flow path through the labyrinth
- \( \rho \) is the density of the fluid mixture
- \( v \) is the flow velocity of the mixture through the channel
- \( C_u \) is a units conversion constant:
  - 1.0E-5 (METRIC)
  - 2.159E-4 (FIELD)
  - 9.869E-7 (LAB)
  - 9.869E-6 (PVT-M).

\( \delta P_{\text{fric}} \) is the standard expression for the homogeneous flow frictional pressure loss through the channel system,

\[ \delta P_{\text{fric}} = 2C_u f \frac{L}{D} \rho v^2 \]  

where

- \( f \) is the Fanning friction factor
- \( D \) is the diameter of each channel
- \( L \) is the effective length of the channel system, which depends on the configuration number.
The grid block connections should be located in the labyrinth segments instead of the segments representing the well tubing. Each labyrinth segment can contain a length of one or more labyrinth device joints, depending on the overall length of the grid block connections that it contains. Note that labyrinth segments should be arranged in parallel, not in series.

The labyrinth segments should be introduced in the normal way using keyword \texttt{WELSEGS}, then defined as labyrinth segments with keyword \texttt{WSEGLABY}. Labyrinth segments should be given a very small length in keyword \texttt{WELSEGS} (of the order, say, of the wellbore radius); this length is not used in any pressure loss calculations, but it will influence the location of the connections in \texttt{COMPSEGS} or \texttt{COMPSEGL}. The labyrinth segments should be given the same depth as their ‘parent’ tubing segments, so that there will be no hydrostatic head across them. The pressure loss sum \( \delta P = \delta P_{\text{flic}} + \delta P_{\text{form}} \) across a labyrinth segment is reported as the friction pressure loss; the acceleration pressure loss is set to zero.

![Figure 3.19. Labyrinth segments branching from the tubing](image)

The \texttt{WSEGLABY} keyword is followed by a number of data records, each containing the following items of data. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the \textit{ECLIPSE Technical Description} for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword \texttt{WLIST}.

   The well must previously have had its segments defined with keyword \texttt{WELSEGS}.

2. Segment number at the start of the range

   An integer between 2 and the \texttt{NSEGMX} mnemonic in the \texttt{WSEGDIMS} keyword.

3. Segment number at the end of the range
An integer between 2 and NSEGMX, not less than item 2.

All segments in this range (that is, with segment numbers between these two values) are treated as labyrinth segments, and the following data items in this record apply to them. To enter data for a single segment, set items 2 and 3 equal.

4. The configuration number, \( N_c \) (an integer)

If the number entered is 0 or negative, the labyrinth pressure drop \( \delta P \) is set equal to zero, as if the device were removed. The total pressure drop along the segment is then the hydrostatic pressure drop only.

Note: A segment with a non-positive configuration number cannot be specified as a secondary link segment in the WSEGLINK keyword.

5. The number of separate channel systems within this segment

The flow through this segment is apportioned equally between the specified number of channel systems. The number of channel systems is related to the number of labyrinth device joints contained within the segment. Normally, each device joint is 12 m long and contains two separate channel systems.

DEFAULT: 2

6. Cross-section area for flow in each channel system

This quantity is used to convert the volumetric flow rate of fluid through each channel system into a flow velocity.

The total cross-section area for flow (that is the product of item 6 and item 5) replaces the area entered in item 9 of keyword WELSEGS.

UNITS: m\(^2\) (METRIC), ft\(^2\) (FIELD), cm\(^2\) (LAB), m\(^2\) (PVT-M)

DEFAULT: 6.0E-5 m\(^2\), 6.46E-4 ft\(^2\), 0.6 cm\(^2\)

7. The base length of the channel path, \( L_1 \)

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: 0.1863 m, 0.6112 ft, 18.63 cm

8. Incremental length of the channel path for each configuration number, \( L_2 \)

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: 0.2832 m, 0.9291 ft, 28.32 cm

9. The channel diameter, \( D \)

This quantity is used in calculation of the friction pressure loss. For non-circular channels, use the equivalent diameter (4 x area / perimeter).

The value entered here replaces the diameter entered in item 7 of keyword WELSEGS.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: 6.316E-3 m, 2.072E-2 ft, 0.6316 cm

10. The absolute roughness of the channel wall
This quantity is used to calculate the Fanning friction factor in the friction pressure loss. The value entered here replaces the roughness entered in item 8 of keyword WELSEGS.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT–M)

DEFAULT: 1.0E-5 m, 3.281E-5 ft, 1.0E-3 cm

11. The dimensionless form loss coefficient for the inlet, $\gamma_{\text{inlet}}$

DEFAULT: 0.35

12. The dimensionless form loss coefficient for the outlet, $\gamma_{\text{outlet}}$

DEFAULT: 0.5

13. Incremental form loss coefficient for each configuration number, $\gamma_{\text{labyr}}$

DEFAULT: 0.55

14. Flag to indicate whether the device is open or shut

OPEN ICD is open and flow is allowed through the device

SHUT ICD is shut and flow is not allowed through the device. Crossflow, if enabled, will still be allowed in the branch.

DEFAULT: OPEN

End the data record with a slash (/).

End the set of data records with a slash.

**Notes**

- A segment can only represent a single device at any one time. Any previous devices generated with the WSEGFLIM, WSEGVALV, WSEGPULL, WSEGSICD and WSEGAICD keywords will be replaced.

- Because the device pressure drop is reported using the segment’s friction pressure drop, friction effects must be included in any multisegment well for which this keyword is specified (see item 6 of WELSEGS).

**Example**

```
WSEGLABY
-- Well Segments Nc Chnis Area L1 L2 Diam Rough Cin Cou Claby
-- Name From To
PROD1 23 23 8 /
PROD1 24 24 6 /
PROD1 25 25 4 4 /
/
```

**Keywords**
WSEGLINK  Defines the chord segment links for specifying looped flow paths

This keyword is used to specify any chord segment links required to form looped flow paths in multisegment wells. (See "Multisegment wells" in the ECLIPSE Technical Description.)

These chord segment links cause the nodes of two specified segments to have the same multisegment well solution variable values (segment pressure and phase flowing fractions in ECLIPSE 100, segment pressure and molar densities in ECLIPSE 300). One of these segments must be at the end of a branch.

The two segments involved are called the primary link segment and the secondary link segment. By default, the first segment specified in the pair will be the primary link segment, but it will be the secondary link segment if it is the only segment in the pair that is at the end of a branch.

The segment structure of any multisegment wells which have chord segment links specified using this keyword must have already been specified using the WELSEGS keyword.

The maximum number of chord segment links that can be specified for a particular well is specified in item 4 of the WSEGDIMS keyword.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name
2. Segment number of the first segment in the chord segment link
   DEFAULT: Undefined
3. Segment number of the second segment in the chord segment link
   DEFAULT: Undefined

*Note:* Each new instance of the WSEGLINK keyword will add to the existing list of chord segment links. If one of the segment numbers is defaulted then all the segment links associated with the other segment in the named well will be deleted. If both the segments are defaulted, then all segment links associated with the named well will be deleted.

*Note:* Any multisegment well for which this keyword is specified must have friction effects included in the segment pressure drop calculation (see item 6 of WELSEGS).

See "Restrictions" in the ECLIPSE Technical Description for a list of restrictions on which segments can be primary and secondary segments in a chord link.

**Example**

In this example, the first record deletes any existing chord segment links in well PROD, the second record deletes any existing chord segment links which involve segment 18 of well INJ1, and the third record creates a chord segment link between segments 22 and 31 of well PROD.

```
WSEGLINK
  PROD          /
  INJ1  18      /
```
This keyword specifies a scaling factor by which the frictional pressure drop calculated across a segment is multiplied, in a multisegment well (see "Multisegment wells" in the ECLIPSE Technical Description). The scaling factor may either be a constant or vary as a function of the water-oil ratio and gas-oil ratio of the mixture flowing through the segment’s outlet junction. A variable scaling factor may be applied, for example, to progressively choke off a branch as its produced gas-oil ratio increases, or to model the operation of an adjustable flow control device whose purpose is to cut off production from completions with high water cuts.

The frictional pressure loss calculated over the segment is multiplied by the function

\[
\max \left[ A + B (WOR)^C + D \left( \frac{GOR}{GOR_{\min}} \right)^E, 1.0 \right]
\]

Eq. 3.262

where \( A \) to \( E \) and \( GOR_{\min} \) are coefficients defined in items 4-9.

It is thus possible to select a set of coefficients which, for example, would begin to reduce flow through a segment once its gas-oil ratio increases above a certain value.

Note: If the segment’s pressure drop is obtained from a VFP table (see keyword WSEGTABL), the total pressure drop interpolated from the table will be multiplied by this function.

If the segment uses the Drift Flux Flow Model (see "The drift flux slip model" in the ECLIPSE Technical Description), the phase ratios reflect the in-situ fluid mixture instead of the flowing mixture (the flowing phase ratios could go negative if counter-current flow occurs). In fact, it is best to represent the flow control device by a segment that uses either a VFP table or the homogeneous flow model for its pressure drop calculation.

The keyword is followed by a number of data records, each containing the following items of data. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

   The well must previously have had its segments defined with keyword WELSEGS.

2. Segment number at the start of the range
   An integer between 2 and the NSEGDX mnemonic in the WSEGDIMS keyword.

3. Segment number at the end of the range
   An integer between 2 and NSEGDX, not less than item 2.

   The following data items apply to all the segments in the range. To enter data for a single segment, set items 2 and 3 equal.

4. Value of the constant \( A \) in the multiplying function
   DEFAULT: 1.0

5. Value of the coefficient \( B \) in the multiplying function
6. Value of the power $C$ in the multiplying function
   The power $C$ must be set > 0.0 if the coefficient $B$ is non-zero.
   DEFAULT: 0.0

7. Value of the coefficient $D$ in the multiplying function
   DEFAULT: 0.0

8. Value of the power $E$ in the multiplying function
   The power $E$ must be set > 0.0 if the coefficient $D$ is non-zero.
   DEFAULT: 0.0

9. Value of $GOR_{\text{min}}$ in the multiplying function
   $GOR_{\text{min}}$ must be set > 0.0 if the coefficient $D$ is non-zero.
   UNITS: sm$^3$/sm$^3$ (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm$^3$/sm$^3$ (PVT-M)
   DEFAULT: 0.0

End the data record with a slash (/).

End the set of data records with a slash.

---

**Note:** If both the coefficients $B$ and $D$ are zero, the pressure loss multiplier will take the constant value $A$, which will **not** be subject to a minimum value of 1.0. However, in this case, $A$ must not be set to a negative value. Apart from this case, any of the coefficients $A$, $B$ and $D$ may be set negative, if required.

The value of the segment frictional pressure drop multiplier can be written to the Summary file using the SUMMARY section keywords SPRDM.

See also keyword **WSEGFLIM**, which provides an alternative means of restricting the flow of associated water and gas through a segment.

---

**Example**

```plaintext
WSEGMULT
  -- Name  Segments  A  B  C  D  E  GORmin
    PROD1   1  6   2.0 /
    PROD1   7  7 -9.0 10.0 2.0 /

/   PROD1

/   PROD1
```

---

**Keywords**

2700
**WSEGPROP**  
Modifies properties of well segments

This keyword may be used to modify properties of individual segments in a multisegment well (see "Multisegment wells" in the ECLIPSE Technical Description) during the simulation, as an alternative to re-entering the segment data with keyword WELSEGS.

The keyword is followed by a number of data records, each containing the following items of data. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

1. Well name, well name template, well list or well list template
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

   The well must previously have had its segments defined with keyword WELSEGS.

2. Segment number at the start of the range
   
   An integer between 2 and the NSEGMX mnemonic in the WSEGDIMS keyword.

3. Segment number at the end of the range
   
   An integer between 2 and NSEGMX, not less than item 2.

   The following segment properties in this record apply to all the segments in the range. To enter properties for a single segment, set items 2 and 3 equal.

4. Tubing internal diameter (or equivalent diameter for annular cross-section)
   
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

   DEFAULT: unchanged from previous value

5. The effective absolute roughness of the tubing
   
   This quantity is used to calculate the Fanning friction factor.

   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

   DEFAULT: from previous value

6. Cross-sectional area for fluid flow
   
   UNITS: m² (METRIC), ft² (FIELD), cm² (LAB), m² (PVT-M)

   DEFAULT: \( \pi (D^2/4) \) if diameter is set in item 4, otherwise unchanged from previous value

7. Volume of the segment
   
   UNITS: m³ (METRIC), ft³ (FIELD), cc (LAB), m³ (PVT-M)

   DEFAULT: \( A \cdot L \) if diameter is set in item 4, otherwise unchanged from previous value

8. **Thermal Option only:** Cross-sectional area of the pipe wall (see note 3 of keyword WELSEGS). For tubing with circular cross-sections the area can be calculated using the formula \( A = \pi \cdot (r_o^2 - r_i^2) \) where \( r_o \) is the outer radius of and \( r_i \) is the inner radius.

   UNITS: m² (METRIC), ft² (FIELD), cm² (LAB), m² (PVT-M)
DEFAULT: Unchanged from previous value.

9. **Thermal Option only**: Volumetric heat capacity of the pipe wall.
   UNITS: kJ/m$^3$/K (METRIC), Btu/ft$^3$/°R (FIELD), J/cc/K (LAB), kJ/m$^3$/K (PVT-M)
   DEFAULT: Unchanged from previous value.

10. **Thermal Option only**: Thermal conductivity of the pipe wall.
    UNITS: kJ/m/day/K (METRIC), Btu/ft/day/°R (FIELD), J/cm/hr/K (LAB), kJ/m/day/K (PVT-M)
    DEFAULT: Unchanged from previous value.

End the data record with a slash (/).

End the set of data records with a slash.

Example

<table>
<thead>
<tr>
<th>WSEGPROP</th>
<th>Name</th>
<th>Segments</th>
<th>Diam</th>
<th>Rough</th>
<th>Area</th>
<th>Vol</th>
</tr>
</thead>
<tbody>
<tr>
<td>PROD1</td>
<td>15</td>
<td>17</td>
<td>0.25</td>
<td></td>
<td>/</td>
<td></td>
</tr>
<tr>
<td>PROD2</td>
<td>12</td>
<td>12</td>
<td>0.2</td>
<td></td>
<td>/</td>
<td></td>
</tr>
</tbody>
</table>

/
WSEGPULL  Defines a segment to represent a pullthrough pump for a down-hole separator

This keyword designates a well segment to represent a pullthrough pump for a downhole water separator in a multisegment well. A pullthrough pump can be used to control the rate of water removal from the separator and its reinjection into the formation. The pump should be situated in the water offtake segment of the separator (item 3 of keyword WSEGSEP) or anywhere further along the water injection branch. For further information on downhole separators, see "Down-hole separators (ECLIPSE 100)" in the ECLIPSE Technical Description.

The pressure differential across the device is the difference of two components:

\[ \delta P = \delta P_{pump} - \delta P_{reduc} \]

\[ \delta P_{pump} \text{ represents the pressure boost from a pump working at a specific net power,} \]

\[ \delta P_{pump} = \min\left(C_u \frac{\text{Pow}_{max}}{V}, \delta P_{max}\right) \]

\[ C_u \text{ is a units conversion constant, given in table 3.54} \]

\[ \text{Pow}_{max} \text{ is the maximum net power of the pump (set in item 3)} \]

\[ V \text{ is the local volumetric flow rate of fluid through the pump} \]

\[ \delta P_{max} \text{ is a maximum pressure boost (set in item 4).} \]

If the pump power is too low, the flow rate of water along the offtake branch may not be able to keep up with the flow of water from the formation into the separator. On the other hand, if the pump power is too high the flow of fluid along the offtake branch may exceed the flow of water from the formation into the separator. In this case the flow in the outlet branch also includes some of the oil from the formation, which is reinjected along with the water. For optimum separation, the flow in the outlet branch should be maintained at a rate that matches the inflow of water into the separator. The pressure reduction term is designed to reduce the pump pressure boost when an excessive amount of oil is carried over into the water injection stream. It reduces the pressure boost by an amount that increases sharply with the excess oil flow rate in the water outlet branch:

\[ \delta P_{reduc} = F(Q_o - Q_{o \text{ lim}}), \quad \text{when} \quad (Q_o > Q_{o \text{ lim}}) \]

\[ F \text{ is a gradient factor (set in item 6)} \]

\[ Q_o \text{ is the flow rate of oil through the device, from the water outlet of the separator} \]

\[ Q_{o \text{ lim}} \text{ is the oil rate limit that triggers the pressure reduction (set in item 5).} \]

To achieve optimum separation, therefore, the pump power and maximum pressure boost should be at least sufficient to reinject all the produced water and the gradient factor in item 6 should be set to a value that enables \( \delta P_{reduc} \) to cancel out \( \delta P_{pump} \) over an acceptable range of oil carry-over rates. A large gradient factor may hinder the convergence of the multisegment well; you are recommended to use WSEGITER to assist the convergence.
The actual applied power of the pump, \( \delta P \cdot V / C_u \), can be written to the Summary file with the SUMMARY section keyword `SPPOW`.

<table>
<thead>
<tr>
<th>Unit system</th>
<th>( C_u )</th>
<th>power</th>
<th>flow rate</th>
<th>pressure</th>
</tr>
</thead>
<tbody>
<tr>
<td>METRIC</td>
<td>864.0</td>
<td>kW</td>
<td>m(^3)/day</td>
<td>bar</td>
</tr>
<tr>
<td>FIELD</td>
<td>3.301E5</td>
<td>hp</td>
<td>ft(^3)/day</td>
<td>psi</td>
</tr>
<tr>
<td>LAB</td>
<td>3.553E4</td>
<td>W</td>
<td>cc/hr</td>
<td>atm</td>
</tr>
<tr>
<td>PVT-M</td>
<td>852.7</td>
<td>kW</td>
<td>m(^3)/day</td>
<td>atm</td>
</tr>
</tbody>
</table>

Table 3.54: Conversion constant in each unit system

The `WSEGPULL` keyword is followed by a number of data records, each containing the following items of data. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing only a slash.

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword `WLIST`.
   The well must previously have had its segments defined with keyword `WELSEGS`.

2. Segment number to contain the pullthrough pump.
   An integer between 2 and the `NSEGMX` mnemonic in the `WSEGdims` keyword.
   The pump should be situated in the water offtake segment of the separator (item 3 of keyword `WSEGSEP`) or anywhere further along the water injection branch.

3. Maximum pump power
   UNITS: kW (METRIC), hp (FIELD), W (LAB), kW (PVT-M)

4. Maximum pressure boost, \( \delta P_{\text{max}} \)
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

5. Limiting rate of oil carryover from the water offtake through the pump, \( Q_{o,\text{lim}} \)
   UNITS: sm\(^3\)/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm\(^3\)/day (PVT-M)
   DEFAULT: 0.0

6. Gradient factor, \( F \), for the pressure reduction term
   UNITS: bars/(sm\(^3\)/day) (METRIC), psi/(stb/day) (FIELD), atm/(scc/hr) (LAB), atm/(sm\(^3\)/day) (PVT-M)
   DEFAULT: 0.0

End the data record with a slash (/).
End the set of data records with a slash.
Note: A segment can only represent a single device at any one time. Any previous devices generated with the \texttt{WSEGLABY}, \texttt{WSEGFLIM}, \texttt{WSEGVALV}, \texttt{WSEGSICD} and \texttt{WSEGAICD} keywords will be replaced.

Example

```
WSEGPULL
   -- Well   Seg   Max   Max   Qo   F
   -- Name   No   Pow   Dp   Lim
   PROD1   23   10.0   300   0.0  50 /
```

Keywords

2705
WSEGSEP  Defines a segment to represent a down-hole separator

A down-hole separator can separate water or free gas from the flowing mixture within a multisegment well (see "Multisegment wells" in the ECLIPSE Technical Description), and send it along a lateral branch to reinject into the formation. This keyword designates a well segment as a downhole separator. The segment must have three segments connecting with it:

- An inlet segment, from which the well stream enters the separator.
- A water or gas offtake segment, which should be the beginning of the lateral branch that injects the separated fluid back into the formation.
- An outlet segment, through which the remaining fluid flows to continue its journey towards the wellhead; this should normally be the separator segment’s outlet segment on the same branch.

The offtake segment, or any other segment on the offtake branch, may contain a pump to control the flow rate of fluid through the offtake. This pump should be modeled either by a VFP table with negative flow values (since the flow is in the direction away from the wellhead) or, for water separation, by a pull-through pump device model (keyword WSEGPULL). A pump may also be placed elsewhere in the well, but not in the separator segment itself.

For a more detailed description of the downhole separator model, please refer to "Down-hole separators (ECLIPSE 100)” in the ECLIPSE Technical Description.

The WSEGSEP keyword is followed by a number of data records, each containing the following items of data. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

   The well must previously have had its segments defined with keyword WELSEGS.

2. Segment number of the down-hole separator segment
   An integer between 2 and the NSEGMX mnemonic in the WSEGDIMS keyword.

3. Segment number of the offtake segment
   An integer between 2 and NSEGMX .

   The offtake segment should be situated on a lateral branch and must connect with the separator segment. The separator segment must have been designated as the outlet segment of the offtake segment in WELSEGS, since the separator should be the offtake’s neighbor in the direction of the wellhead. Note that the actual flow is normally in the opposite direction.

4. The separated phase that the offtake preferentially removes
   WAT  The offtake preferentially removes water from the fluid in the separator
   GAS  The offtake preferentially removes free gas (at the separator segment’s pressure) from the fluid in the separator
The offtake acts as a standard outlet/inlet to the segment in item 2, which ceases to act as a downhole separator.

DEFAULT: NONE

5. The maximum separation efficiency for the offtake, $E_{\text{max}}$

A number between 0.0 and 1.0

The holdup fraction, $\alpha_{\text{off}}$, of the offtake’s preferential phase is equal to this value, as long as the phase’s holdup fraction within the remaining mixture in the separator segment keeps above the value specified in item 6.

DEFAULT: 1.0

6. The limiting holdup fraction, $\alpha_{\text{lim}}$, below which the separation efficiency begins to decrease

When the phase’s holdup fraction remaining in the separator segment, $\alpha_{\text{sep}}$, lies below the value $\alpha_{\text{lim}}$, the separation efficiency decreases proportionally towards zero, $E = E_{\text{max}} \left( \frac{\alpha_{\text{sep}}}{\alpha_{\text{lim}}} \right)$.

The value of $\alpha_{\text{lim}}$ must be greater than zero and must not exceed 1.0. The smaller the value is, the smaller the fraction of the separated phase remaining is within the separator to flow towards the wellhead. However, very small values (< .001) may result in convergence difficulties.

DEFAULT: 0.01

End the data record with a slash (/).

End the set of data records with a slash.

**Note:** The separator segment should use the homogeneous flow model. If it has previously been set up to use the drift flux model, ECLIPSE automatically changes it to homogeneous flow.

The operation of a down-hole separator may increase the number of well model iterations required to converge the well solution. In cases of convergence difficulty, use keyword **WSEGITER** to increase the number of iterations and/or iteration cycles.

---

### Example

```
WSEGSEP

-- Well  Separator  Offtake  Offtake  Maximum  Limiting
-- Name  Segment  Segment  Phase  Efficiency  Holdup Fr
PROD1  3  14  'WAT'  1.0  0.001 /
```

---

**Keywords**

2707
WSEGSICD  Defines segments to represent a spiral inflow control device

This keyword designates specific well segments to represent a spiral inflow control device (ICD) in a multisection well (see “Multisegment wells” in the ECLIPSE Technical Description). This type of device is used to control the inflow profile along a horizontal well or branch by imposing an additional pressure drop between the sand face and the tubing. The device is placed around a section of the tubing and diverts the fluid inflowing from the adjacent part of the formation through a sand screen and then into a spiral before it enters the tubing.

The pressure drop across the device is calculated from calibration data, adjusted to allow for the varying density and viscosity of the reservoir fluid flowing through the device:

\[
\delta P = \left( \frac{\rho_{cal}}{\rho_{mix}} \cdot \frac{\mu_{mix}}{\mu_{cal}} \right)^{1/4} \cdot \frac{\rho_{mix}}{\rho_{cal}} \cdot K \cdot q^2
\]

Eq. 3.266

where

- \( \rho_{mix} \) is the density of the fluid mixture in the segment at local conditions
- \( \rho_{cal} \) is the density of the fluid used to calibrate the ICD (see item 6)
- \( \mu_{mix} \) is the viscosity of the fluid mixture in the segment at local conditions
- \( \mu_{cal} \) is the viscosity of the fluid used to calibrate the ICD (see item 7)
- \( K \) is the base strength of the ICD, with dimension of inverse area squared, defined as:

\[
K = \frac{a_{SICD}}{\rho_{cal}}
\]

Eq. 3.267

where

- \( a_{SICD} \) is defined as the strength of the SICD (see item 4)
- \( q \) is the volume flow rate of fluid mixture through the ICD at local conditions, which is equal to the volume flow rate through the ICD segment multiplied by a scaling factor that depends on the length of the device (see item 5).

The density of the fluid mixture at local segment conditions is given by:

\[
\rho_{mix} = \alpha_o \cdot \rho_o + \alpha_w \cdot \rho_w + \alpha_g \cdot \rho_g
\]

Eq. 3.268

where

- \( \alpha_{o,w,g} \) is the volume fraction of the free oil, water, gas phases at local conditions
- \( \rho_{o,w,g} \) is the density of the oil, water, gas phases at local conditions.

The viscosity of the fluid mixture at local segment conditions is given by:
\[ \mu_{\text{mix}} = (\alpha_o + \alpha_w) \cdot \mu_{\text{emul}} + \alpha_g \cdot \mu_g \]  

Eq. 3.269

where

\( \mu_{\text{emul}} \) is the viscosity of the oil-water emulsion at local conditions

\( \mu_g \) is the gas viscosity at local conditions.

The calculation of \( \mu_{\text{emul}} \) is described in "Emulsion viscosity".

To include a series of these devices in a multisegment well, the devices should be represented by segments branching off the tubing as shown in figure 3.20. The grid block connections should be located in the ICD segments instead of the segments representing the well tubing.

The ICD segments should be introduced in the normal way using keyword WELSEGS, then defined as ICD segments with keyword WSEGSICD. ICD segments should be given a very small length in keyword WELSEGS (of the order, say, of the wellbore radius); this length is not used in any pressure loss calculations, but it will influence the location of the connections in COMPSEGS or COMPSEGL. The ICD segments should be given the same depth as their ‘parent’ tubing segments, so that there will be no hydrostatic head across them. The pressure loss across an ICD segment is reported as the friction pressure loss; the acceleration pressure loss is set to zero.

![Diagram](image)

Figure 3.20. ICD segments branching from the tubing

The figure shows a typical configuration of ICD segments. If, however, you wish to define a more complicated configuration where the flow may pass through more than one ICD segment you must declare the ICD segments in WSEGSICD in order with upstream segments (that is those further from the wellhead) declared before downstream segments.

The WSEGSICD keyword is followed by a number of data records, each containing the following items of data. Unless otherwise stated, entering a negative value for an item defaults the item. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

1. Well name, well name template, well list or well list template.

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

   The well must previously have had its segments defined with keyword WELSEGS.

2. Segment number at the start of the range.

   An integer between 2 and the argument NSEGMX in the WSEGDIMS keyword.
3. Segment number at the end of the range
   An integer between 2 and NSEGMAX, not less than item 2.
   All segments in this range (that is, with segment numbers between these two values) are treated as
   ICD segments, and the following data items in this record apply to them. To enter data for a single
   segment, set item 2 and item 3 equal.

4. The strength of the SICD, \(a_{SICD}\)
   This is an empirical constant based on measurements of the calibrating fluid flow through a particular
   type of ICD.
   UNITS: bars/(\(\text{m}^3/\text{day}\))^2 (METRIC), psi/(\(\text{ft}^3/\text{day}\))^2 (FIELD), atm/(\(\text{rrc/hr}\))^2 (LAB), atm/(\(\text{m}^3/\text{day}\))^2
   (PVT-M)

5. The length of the ICD, \(l_{ICD}\)
   This value is used (together with item 11) to determine a scaling factor which is used to scale the flow
   from the reservoir in order to establish the flow, \(q_{ICD}\), through each ICD. The calculation is described
   in “Flow scaling”.
   There are three possibilities for calculation of the flow scaling factor:
   a. The scale factor is equal to the length of the ICD, \(l_{ICD}\), divided by the length of the tubing
      segment that is the parent of the ICDs to account for the fact that the ICD segment may represent
      a number of ICDs in parallel. (item 11 = 0)
   b. The scale factor is equal to the absolute value of \(l_{ICD}\). (item 11 = 1)
   c. The scale factor is equal to the length of the ICD, \(l_{ICD}\), divided by the total length of the
      completions which supply the ICD. (item 11 = 2)
      item 11 allows you to specify explicitly which of these options is used. If item 11 is defaulted,
      then option a) is used whenever \(l_{ICD}\) is positive, and option b) when \(l_{ICD}\) is negative.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   DEFAULT: 12 m, 39.37 ft, 1,200 cm

6. The density of the calibration fluid, \(\rho_{cal}\)
   UNITS: kg/m^3 (METRIC), lb/\(\text{ft}^3\) (FIELD), gm/cc (LAB), kg/m^3 (PVT-M)
   DEFAULT: 1000.25 kg/m^3, 62.416 lb/\(\text{ft}^3\), 1.00025 gm/cc

7. The viscosity of the calibration fluid, \(\mu_{cal}\)
   UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)
   DEFAULT: 1.45 cP

8. The critical value of the local water in liquid fraction used to select between a water-in-oil or oil-in-
   water equation for the emulsion viscosity; this is described in the “Emulsion viscosity”.
   UNITS: dimensionless
   DEFAULT: 0.5
9. The width of the transition region about the critical value of the local water in liquid fraction (see item 8). For water in liquid fractions within this region, the emulsion viscosity is a linear interpolation between the water-in-oil and oil-in-water viscosity values either side of the region. This transition region is needed to ensure that viscosity is continuous function of water in liquid fraction as described in the "Emulsion viscosity".

UNITs: dimensionless
DEFAULT: 0.05

10. The maximum permitted value of the ratio of the emulsion viscosity to continuous phase viscosity. This usually only affects the water-in-oil viscosity and results in a 'plateau' region within which the emulsion viscosity is at its maximum permitted value as shown in figure 3.21. In this figure, the plateau is bounded by the transition between oil continuous and water continuous emulsions (see item 8). This is described in the "Emulsion viscosity".

UNITs: dimensionless
DEFAULT: 5.0

11. An integer which determines how the flow scaling factor is calculated.

0 The flow scale factor is equal to the ICD length (item 5) divided by the length of the tubing segment that is the parent of the ICD device, which is taken directly from the WELSEGS keyword (case a)

1 The flow scale factor is equal to the absolute value of item 5 (case b)

2 The flow scale factor is equal to the ICD length (item 5) divided by the total length of all completions which feed into the ICD (case c).

If this integer is set to a negative value, then the pre-2008.2 behavior is obtained. That is, if item 5 is positive then the flow scaling factor is equal to the ICD length divided by the length of the encased tubing segment, and if item 5 is negative then the flow scaling factor is the absolute value of item 5.

DEFAULT: -1

12. The maximum absolute value of $q_{ICD}$ for which equation 3.266 holds. This value will typically be the maximum flow-rate for which the device was calibrated. Above this value the equation will be extrapolated linearly.

UNITs: m$^3$/day (METRIC), ft$^3$/day (FIELD), cc/hr (LAB), m$^3$/day (PVT-M)

DEFAULT: No limit set

13. Flag to indicate whether the device is open or shut.

OPEN ICD is open and flow is allowed through the device

SHUT ICD is shut and flow is not allowed through the device. Crossflow, if enabled, will still be allowed in the branch.

DEFAULT: OPEN

End the data record with a slash (/).
End the set of data records with a slash.
**Emulsion viscosity**

The emulsion viscosity is a function of the local phase volume fractions in the segment and has differing functional forms at low water in liquid fractions (when oil is the continuous phase) and high water in liquid fractions (when water is the continuous phase). A critical water in liquid fraction (see item 8) is used to select between these equations given below ([Ref. 77] and [Ref. 114] in the ECLIPSE Technical Description):

\[
\mu_{wio} = \mu_o \cdot \left( \frac{1}{1 - \left( \frac{0.8415}{0.7480} \cdot \alpha_{wl} \right)^{2.5}} \right) \quad \text{Eq. 3.270}
\]

\[
\mu_{oiw} = \mu_w \cdot \left( \frac{1}{1 - \left( \frac{0.6019}{0.6410} \cdot \alpha_{ol} \right)^{2.5}} \right) \quad \text{Eq. 3.271}
\]

where

- \( \mu_{wio} \) is the water-in-oil emulsion viscosity (when oil is the continuous phase)
- \( \mu_{oiw} \) is the oil-in-water emulsion viscosity (when water is the continuous phase)
- \( \mu_o \) is the oil viscosity at local conditions
- \( \mu_w \) is the water viscosity at local conditions
- \( \alpha_{wl} \) is the local water in liquid fraction, \( = \frac{\alpha_w}{\alpha_w + \alpha_o} \)
- \( \alpha_{ol} \) is the local oil in liquid fraction, \( = \frac{\alpha_o}{\alpha_w + \alpha_o} \)

The water-in-oil viscosity is subject to an upper limit expressed as a maximum ratio of water-in-oil viscosity to oil viscosity (see item 10). This usually results in a ‘plateau’ region within which the water-in-oil viscosity is at its maximum permitted value as shown schematically in figure 3.21, with the maximum viscosity ratio set at 5.0. This upper limit also applies to the oil-in-water viscosity, but is less commonly encountered.

At the critical water in liquid fraction there is a jump in emulsion viscosity as the continuous phase changes. Such a discontinuity would cause stability problems in the simulator and a transition region is defined about the critical water in liquid fraction to avoid this. In this region the emulsion viscosity is linearly interpolated between the water-in-oil and oil-in-water viscosities at the edges of the region; the viscosity is thus a continuous function of the water in liquid fraction. The width of the transition region is defined in item 9 above. This transition region is shown schematically in figure 3.21, with the linear interpolation shown in red between points B and C.
The following rules are used to obtain an emulsion viscosity for a given local water in liquid fraction:

- For local water in liquid fractions below the critical value and outside the transition region (A to B) the water-in-oil emulsion viscosity equation, equation 3.270 is used, subject to a maximum permitted viscosity ratio supplied in item 10.
- For local water in liquid fractions within the transition region about the critical value (B to C), the emulsion viscosity is a linear interpolation between the water-in-oil viscosity calculated at the continuous oil region edge and the oil-in-water viscosity calculated at the continuous water region edge.
- For local water in liquid fractions above the critical value and outside the transition region (C to D) the oil-in-water emulsion viscosity equation, equation 3.271 is used, subject to a maximum permitted viscosity ratio supplied in item 10.

**ECLIPSE 100 only**

The emulsion viscosity model may be disabled by setting item 88 to 1 or 3 in the OPTIONS keyword. Note that the value of 3 also activates the emulsion viscosity model for the WSEGAICD keyword.

**ECLIPSE 300 only**

The emulsion viscosity model may be disabled by setting item 169 to 1 or 3 in the OPTIONS3 keyword. Note that the value of 3 also activates the emulsion viscosity model for the WSEGAICD keyword.

The mixture viscosity used in equation 3.266 becomes a local volume fraction weighted sum of each phase’s viscosity:

$$\mu_{mix} = a_o \cdot \mu_o + a_w \cdot \mu_w + a_g \cdot \mu_g$$

Eq. 3.272

**Flow scaling**

An ICD segment may represent a number of ICDs in parallel. Thus the flow rate passing through each ICD is equal to the flow rate through the segment divided by the number of ICDs it represents.
To estimate the number of ICDs in parallel contained in an ICD segment, we compare the length of a single ICD with the length of the tubing segment that the devices encase. We take this tubing segment to be the ICD segment’s immediate neighbor towards the wellhead, that is, its outlet segment as shown in figure 3.20. This assumes that the tubing is encased by a contiguous series of ICDs, into which the fluid from the formation flows.

Thus the segment flow is scaled as follows:

\[ q_{ICD} = q_{seg} \times \frac{l_{ICD}}{l_{tubing}} \]  
Eq. 3.273

where

- \( q_{ICD} \) is the scaled volume flow rate to be used in the pressure drop equation 3.266
- \( q_{seg} \) is the volume flow rate from the reservoir into the ICD segment, which is equal to the actual flow rate through the segment
- \( l_{ICD} \) is the length of an actual ICD (see item 5)
- \( l_{tubing} \) is the length of the tubing segment that is the parent of the ICD segment. This length is taken directly from the WELSEGS keyword.

This flow scaling equation 3.273 is still used if the tubing segment length is smaller than the ICD length. A non-integer scaling factor will be applied if the tubing segment is not an exact multiple of the ICD length (that is the scaling is not adjusted to the nearest integer).

Alternatively, the number of ICDs in parallel contained in an ICD segment can be estimated by comparing the length of the ICD with the total length of all completions which feed it. The length of all completions on the ICD segment and on any upstream segment (including branches) are included in the total length. However, if any other devices (as generated by the WSEGSICD, WSEGLABY, WSEGFLIM, WSEGVALV, WSEGPULL, WSEGAICD and WSEGTABL keywords) are present on upstream segments, only completions encountered prior to these devices are included.

Thus the segment flow is scaled as follows:

\[ q_{ICD} = q_{seg} \times \frac{l_{ICD}}{l_{comp}} \]  
Eq. 3.274

where

- \( q_{ICD} \) is the scaled volume flow rate to be used in the pressure drop equation 3.266
- \( q_{seg} \) is the flow rate through the ICD segment
- \( l_{ICD} \) is the length of an actual ICD (see item 5)
- \( l_{comp} \) is the total length of all completions upstream of the ICD and hence which feed into it.

For more complicated circumstances, the scaling calculations above may not be appropriate. If this is the case you can enter the scaling factor manually into item 5 (either by setting both items 5 and 11 to negative values or setting item 11 to 1). It will not then be interpreted as the ICD length, but its absolute value directly multiplies the segment flow rate, \( q_{seg} \), to give the flow rate used in the pressure drop, \( q_{ICD} \).
Note: A segment can only represent a single device at any one time. Any previous devices generated with the WSEGLABY, WSEGFLIM, WSEGVALV, WSEGPULL and WSEGAICD keywords will be replaced.

Note: Because the device pressure drop is reported via the segment’s friction pressure drop, friction effects must be included in any multisegment well for which this keyword is specified (see item 6 of WELSEGS).

Example

<table>
<thead>
<tr>
<th>WSEGSICD</th>
<th>--Well Segment ICD ICD dens visc crit width max scal</th>
</tr>
</thead>
<tbody>
<tr>
<td>--Name From To stren length cal cal local trans visc fact</td>
<td></td>
</tr>
<tr>
<td>-- 1d 1d 1d 1d 1d 1d 1d 1d 1d</td>
<td></td>
</tr>
<tr>
<td>PROD1 23 23 0.00021 1* 1.0 0.45 0.60 1* 1* 1*/</td>
<td></td>
</tr>
<tr>
<td>PROD1 24 24 0.00021 1* 1.0 0.50 0.65 0.02 1* 1*/</td>
<td></td>
</tr>
<tr>
<td>PROD1 25 25 0.00042 15 1.0 0.45 0.50 1* 6 1*/</td>
<td></td>
</tr>
</tbody>
</table>

/
**WSEGSOLV** Sets parameters affecting the multisegment well iterative linear solver

This keyword controls the iterative linear solver that is used for multisegment wells that contain segment chord links, that is, loops specified using the `WSEGLINK` keyword. (See "Multisegment wells" in the *ECLIPSE Technical Description*). This keyword also allows this solver to be applied to multisegment wells without loops.

Due to the additional matrix terms caused by the additional flowpaths in looped wells, additional storage would be required while solving the well’s linear system within each Newton iteration, so to avoid this additional storage requirement, these wells are solved using an iterative linear solver.

This iterative solver uses an incomplete LU factorization (ILU) pre-conditioner. You can specify how close this incomplete factorization approximates the true LU factorization using the fill level and drop tolerance that are items 4 and 5 of this keyword respectively.

For wells without loops, as long as the segments are ordered carefully, there is no fill-in when calculating the LU-factorization of the well linear system (that is, no additional non-zero entries are generated). Hence a direct solver, in particular a technique known as the “cheap LU factorization”, is used. However, the default implementation has some inefficiencies so the memory requirement and computational expense is higher than strictly necessary. For this reason, it can be preferable to choose the multisegment well iterative solver, which should solve the well linear system directly (and efficiently) purely in the pre-conditioning step without the need for iterations. If the multisegment well iterative linear solver is used for a well with no loops, then the “cheap LU factorization” form of the ILU pre-conditioner is automatically used.

The `WSEGSOLV` keyword is followed by a data record consisting of the following items, terminated with a slash (/).

1. Whether to use the iterative linear solver for wells which do not have any segment chord links (see the `WSEGLINK` keyword).
   - **YES** Use the iterative solver for all multisegment wells, even if they do not have a chord segment link. (This can reduce the memory requirement for runs with large numbers of segments per well.)
   - **NO** Only use the iterative solver for multisegment wells which have a chord segment link.
     
     **DEFAULT:** NO

2. Maximum number of iterations of the linear solver when solving the multisegment well linear systems
   
   **DEFAULT:** 30

3. Target convergence tolerance in the multisegment well iterative linear solver. This is the absolute target for the L2-norm of the well equation residuals.
   
   **DEFAULT:** 1.0E-10

4. Fill level of the ILU pre-conditioner in linear solver. This indicates how much fill-in is allowed when calculating the incomplete LU-factorization of the well linear system - this incomplete factorization is used as a pre-conditioner in the linear iterations.
   
   The higher the level of fill, the greater the memory requirement for the pre-conditioner.
   
   **DEFAULT:** 2
5. Drop tolerance level in the ILU pre-conditioner in linear solver. In the LU decomposition of the well linear system, any fill-in values which would have an absolute value below this drop tolerance are ignored. This value has no effect if the fill level specified in item 4 is zero.

The lower the drop tolerance, the greater the memory requirement for the pre-conditioner.

DEFAULT: 0.0

6. Limit beneath which the pressure drop derivative with respect to total flow is linearized for segments which represent ICDs. This is used to avoid a zero derivative at low flows which can cause problems for the iterative linear solver. This item only affects secondary linked segments containing devices specified with the WSEGAICD, WSEGSICD, WSEGLABY or WSEGVALV keywords. If you increase the number of iteration cycles or reduction factor in the WSEGITER keyword you may find it is also necessary to increase this number.

DEFAULT: 1.0E-8

ECLIPSE 100 only 7. Looped well flow chop threshold.

During the non-linear solving of looped multisegment wells, spurious high flows are occasionally generated. Using items 7 and 8 of WSEGSOLV, these flows can be controlled. Any attempt by the solver to increase a segment total flow control variable from an absolute value above the threshold (item 7) by a factor greater than the multiplier (item 8) will be chopped to the maximum allowed value. Negative values disable the flow chops. The total flow control variable is a weighted sum of phase flows. Although optimum values will vary from case to case, a value of 10 for each of items 7 and 8 is reasonable.

UNITS: sm$^3$/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm$^3$/day (PVT-M)

DEFAULT: -1.0

ECLIPSE 100 only 8. Looped well flow chop multiplier. See item 7 for a description.

DEFAULT: -1.0

Example

```
WSEGSOLV
  1* 50 /
```

Keywords

2717
WSEGTABL  Calculate segment pressure drops from VFP tables

This keyword designates specific segments in a multisegment well to have their pressure drops obtained from a VFP table instead of a built-in flow model (see “Pressure drops from VFP tables” in the ECLIPSE Technical Description).

The keyword is followed by a number of data records, each containing the following items of data. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

1.   Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

   The well must previously have had its segments defined with keyword WELSEGS.

2.   Segment number at the start of the range

   An integer between 2 and the NSEGMX mnemonic in the WSEGDIMS keyword.

3.   Segment number at the end of the range

   An integer between 2 and NSEGMX, not less than item 2.

   The following data items in this record will be applied to all the segments in the range. To enter data for a single segment, set items 2 and 3 equal.

4.   Production VFP table number

   A production well VFP table with this number must previously have been entered with keyword VFPPROD. The second item in its first record is interpreted as either its datum length or datum depth span, to allow ECLIPSE to scale the interpolated pressure drop along each segment in proportion to its length or depth span (see item 7). The table may extend into negative flows, which are interpreted as flow in the direction away from the wellhead (positive flows are towards the wellhead). There must be no point at zero flow, since the WFR and GFR values are singular there.

   If the value 0 is entered, the pressure drop calculation reverts to the Homogeneous Flow Model.

   DEFAULT:   0

5.   Components of the pressure drop that are included in the VFP table

   FH  Friction and Hydrostatic.

   The VFP table contains the sum of the friction and hydrostatic pressure drops. The combined pressure drop interpolated from the VFP table is reported in the output as a friction pressure drop, since the table contains no information to resolve it into the two components.

   F-  Friction only.

   The hydrostatic pressure drop is calculated from the fluid density in the segment and added to the friction pressure drop interpolated from the VFP table. This option is useful to apply to undulating sections of tubing, to overcome the need to have a separate table for each inclination angle.
The acceleration pressure drop is calculated separately, and added to the friction and hydrostatic pressure drops, if HFA was entered in item 6 of Record 1 of keyword WELSEGS. It is calculated using the segment’s diameter and area entered in WELSEGS or WSEGPROP. (See "Pressure drop components" in the ECLIPSE Technical Description.)

DEFAULT: FH

6. Procedure for handling flows that are more negative (or less positive) than the first flow value in the VFP table

REV  Reverse the flows and pressure losses in the table.

This procedure is recommended for use in horizontal segments or segments representing chokes, or with friction only VFP tables (see item 5), in which the table’s pressure drop has no hydrostatic component. In these cases the pressure drop can simply be reversed when the flow reverses. The first flow value in the VFP table should be positive. For negative flows, the mirror image of the table is used. For flows in between + and - the first value, the pressure loss is linearly interpolated between these two points.

FI  Fix the lookup value of the flow rate at the first flow point in the table.

This procedure is recommended for use in vertical or inclined segments in which the friction pressure loss in reverse flow is expected to be small compared with the hydrostatic pressure loss. (The REV option is not suitable in these cases as the hydrostatic pressure loss does not reverse when the flow reverses.) If the first flow value in the VFP table is positive, it should be low enough to give a negligible friction pressure drop, so that the calculated friction pressure drop remains negligible during reverse flow.

EXT  Extrapolate the pressure drop linearly with flow.

This should be used with caution, as substantial extrapolation (especially into the negative flow region) may result in stability problems.

DEFAULT: there is no default for this item, as the appropriate choice depends on the type of segment.

Note: For flows that lie above the highest value in the table, the table is extrapolated linearly as normal. For WFR and GFR values that lie outside their range of tabulated values, the lookup value of the respective quantity is fixed at its nearest tabulated point. This is to avoid stability problems that can arise when the table is extrapolated substantially in these quantities.

7. Instruction for scaling the interpolated pressure drop according to the length or depth span of each segment

LEN  The interpolated pressure drop is scaled in proportion to the length of the segment relative to the table’s datum length.

The second item in the first record of VFPPROD is interpreted as the table’s datum length.

DEP  The interpolated pressure drop is scaled in proportion to the depth span of the segment (that is the change in depth over the segment’s length) relative to the table’s datum depth span.

The second item in the first record of VFPPROD is interpreted as the table’s datum depth span.

NO  The interpolated pressure drop is not scaled automatically according to the segment properties. (This option is recommended if the VFP table represents the pressure loss across a choke.)
DEFAULT: LEN

(See "Pressure drop scaling" in the ECLIPSE Technical Description.)

8. ALQ value used in the table look-up. User defined arguments can be set with keywords UDQ and UDADIMS.

UNITs: These depend on what the ALQ was meant to represent when the table was created. Refer to section "Record 6:ALQ values" in the description of keyword VFPPROD. For example, if the table was created by VFP i to model pressure loss across a choke (with an ALQ definition of BEAN), the ALQ represents the choke diameter in units of mm (METRIC), 1/64ths in (FIELD).

DEFAULT: 0.0

9. Flag to indicate whether the segment is open or shut

OPEN  Segment is open and flow is allowed

SHUT  Segment is shut and flow is not allowed through. Crossflow, if enabled, will still be allowed in the branch.

DEFAULT: OPEN

End the data record with a slash (/).

End the set of data records with a slash.

Example

<table>
<thead>
<tr>
<th>Well</th>
<th>Segments</th>
<th>VFP</th>
<th>P Drop</th>
<th>Scale</th>
<th>ALQ</th>
</tr>
</thead>
<tbody>
<tr>
<td>PROD1</td>
<td>5 12 4</td>
<td>FH</td>
<td>FIX</td>
<td>LEN</td>
<td>/</td>
</tr>
<tr>
<td>PROD1</td>
<td>4 4 5</td>
<td>F-</td>
<td>REV</td>
<td>NO</td>
<td>/</td>
</tr>
</tbody>
</table>

End the set of data records with a slash.
WSEGVALV  Defines a segment to represent a sub-critical valve

This keyword designates specific well segments to represent a sub-critical valve in a multisegment well (see "Multisegment wells" in the ECLIPSE Technical Description). This imposes an additional pressure drop in the segment due to flow through a constriction with a specified area of cross-section.

The WSEGVALV keyword is followed by a number of data records, each containing the following items of data. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing only a slash.

1. Well name, well name template, well list or well list template.
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

   The well must previously have had its segments defined with keyword WELSEGS.

2. Segment number to contain the valve.
   An integer between 2 and the NSEGMX argument in the WSEGDIMS keyword.

3. The dimensionless flow coefficient for the valve, Cv

4. Cross-section area for flow in the constriction, Ac
   This quantity is used to convert the volumetric flow rate of fluid through the segment into the flow velocity at the constriction, vc.
   UNITS: m² (METRIC), ft² (FIELD), cm² (LAB), m² (PVT-M)

5. Additional length of pipe for the friction pressure drop, L
   Set this to zero if there is no additional friction pressure loss across this segment.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   DEFAULT: length of the segment as derived from the WELSEGS data

6. The pipe diameter, D, for the frictional pressure drop calculation.
   For non-circular pipes, use the equivalent diameter (4 x area / perimeter).
   The value entered here replaces the diameter entered in item 7 of keyword WELSEGS.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
   DEFAULT: no change; the initial value is taken from item 7 of keyword WELSEGS unless a previous instance of WSEGVALV defined a value.

7. The absolute roughness of the pipe wall.
   This quantity is used to calculate the Fanning friction factor in the friction pressure loss.
   The value entered here replaces the roughness entered in item 8 of keyword WELSEGS.
   UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
The pipe cross-section area, $A_p$, for the frictional pressure drop calculation.

This quantity is used to convert the volumetric flow rate of fluid through the segment into the flow velocity in the pipe, $v_p$.

The value entered here replaces the area entered in item 9 of keyword WELSEGS.

UNITs: $m^2$ (METRIC), $ft^2$ (FIELD), $cm^2$ (LAB), $m^2$ (PVT-M)

DEFAULT: no change; the initial value is taken from item 9 of keyword WELSEGS unless a previous instance of WSEGVALV defined a value.

9. Flag to indicate whether the device is open or shut.

OPEN ICD is open and flow is allowed through the device

SHUT ICD is shut and flow is not allowed through the device. Crossflow, if enabled, will still be allowed in the branch.

DEFAULT: OPEN

10. The maximum cross-sectional area for flow in the constriction, $A_{max}$.

DEFAULT: $A_{max}$ is taken from item 8, if defined, or else from item 9 of keyword WELSEGS.

UNITs: $m^2$ (METRIC), $ft^2$ (FIELD), $cm^2$ (LAB), $m^2$ (PVT-M)

End the data record with a slash (/).

End the set of data records with a slash.

The valve modeling

The pressure drop across the device is calculated using a homogeneous model of sub-critical flow through a pipe containing a constriction:

$$\delta P = \delta P_{cons} + \delta P_{fric}$$

Eq. 3.275

$\delta P_{cons}$ accounts for the effects of the constriction, and is calculated as:

$$\delta P_{cons} = C_u \frac{\rho v_c^2}{2C_v^2}$$

Eq. 3.276

$C_u$ is a units conversion constant, which is given in table 3.55.

$\rho$ is the density of the fluid mixture,

$v_c$ is the flow velocity of the mixture through the constriction,

$C_v$ is a dimensionless flow coefficient for the valve.
\[ \delta P_{\text{fric}} = 2C_u f \frac{L}{D} \rho v_p^2 \]  

Eq. 3.277

\( f \) is the Fanning friction factor

\( L \) is the additional length of piping in the segment

\( D \) is the diameter of the pipe (not the constriction)

\( v_p \) is the flow velocity of the mixture through the pipe.

Obviously \( v_c \) and \( v_p \) depend on the respective cross-section areas of the constriction and the pipe and the local volumetric flow rate \( q_m \) of the mixture through the segment,

\[ q_m = v_c A_c = v_p A_p \]  

Eq. 3.278

The pressure loss sum \( \delta P = \delta P_{\text{fric}} + \delta P_{\text{cons}} \) across a valve segment is reported as the friction pressure loss; the acceleration pressure loss is set to zero.

Using the relationship from equation 3.278 \( \delta P_{\text{cons}} \) can be expressed as:

\[ \delta P_{\text{cons}} = C'_u \rho q_m \left| \frac{q_m}{q_m} \right| \frac{1}{2C^2_v A^2_c} \]  

Eq. 3.279

\( A_c \) is the cross-sectional area of the valve constriction and takes a minimum value of 1.0E-10,

\( C'_u \) is the modified unit conversion factor, which is given in the following table.

<table>
<thead>
<tr>
<th>Unit system</th>
<th>( C'_u )</th>
<th>Density</th>
<th>Volume flow</th>
<th>Pressure</th>
</tr>
</thead>
<tbody>
<tr>
<td>METRIC</td>
<td>1.340E-15</td>
<td>kg/m³</td>
<td>m³/day</td>
<td>bars</td>
</tr>
<tr>
<td>FIELD</td>
<td>2.892E-14</td>
<td>lb/ft³</td>
<td>ft³/day</td>
<td>psi</td>
</tr>
<tr>
<td>LAB</td>
<td>7.615E-14</td>
<td>gm/cc</td>
<td>cc/hour</td>
<td>atm</td>
</tr>
<tr>
<td>PVT-M</td>
<td>1.322E-15</td>
<td>kg/m³</td>
<td>m³/day</td>
<td>atm</td>
</tr>
</tbody>
</table>

Table 3.56: Modified conversion constant in each unit system for the pressure drop across a device

The base strength of the device, \( K \), with dimension of inverse area squared, can be defined from equation 3.279 as:
\[ K = \frac{C_u'}{\left(2C'_vA^2\right)} \]  

Eq. 3.280

If the strength of the valve is greater than 0.1 it will cause the valve to shut. The setting of the device, how open it is relative to the maximum constriction area, is defined as:

\[ \frac{A_c}{A_{\text{max}}} \]  

Eq. 3.281

\( A_{\text{max}} \) is the maximum constriction area.

**Notes**

- A segment can only represent a single device at any one time. Any previous devices generated with the `WSEGLABY`, `WSEGFLIM`, `WSEGPULL`, `WSEGSICD` and `WSEGAICD` keywords will be replaced.

- Because the device pressure drop is reported using the segment’s friction pressure drop, friction effects must be included in any multisegment well for which this keyword is specified (see item 6 of `WELSEGS`).

- When defining a valve, to ensure that the setting of `SUMMARY` keyword SFOPN for the device is accurate for the entirety of the run, the keyword should include the maximum cross-sectional area in the constriction for flow. This is done in item 10, \( A_{\text{max}} \). If no value is given, it will be defaulted to item 8, if defined, otherwise it will be defaulted to item 9 of keyword `WELSEGS`.

**Example**

```
WSEGVALV
-- Well   Seg   Cv   Ac   L   D   Rough   Ap
-- Name   No
PROD1   23   0.66   0.01   0.0 / no friction length
PROD1   24   0.66   0.01 / pipe data taken from WELSEGS /
```
WSEGWELL  Segment secondary well name

This keyword provides the facility to associate a specified segment of a multisegment well with a secondary well name (see "Multisegment wells" in the ECLIPSE Technical Description). When this association has been established, a number of secondary well controls may be applied to the segment via its secondary well name alias. The WSEGWELL keyword is followed by one or more data records to define the well, the segment number, the secondary well name and the group to which it belongs. Each record is terminated with a slash (/), and the set of records must end with an additional blank record containing just a slash.

The data items in each record are:

1. Well name
   
   A character string of up to eight characters in length.
   
   The well must have previously been defined as a multisegment well and had its segments defined with keyword WELSEGS.

2. Segment number to be associated with the secondary well name
   
   An integer between 2 and the NSEGMX mnemonic in the WSEGDIMS keyword. The specified segment must previously have been defined for this well.

3. Secondary well name
   
   A character string of up to eight characters in length.
   
   This must not be a previously defined well name.

4. Group name
   
   A character string of up to eight characters in length.
   
   DEFAULT: The group to which the well defined in item 1 currently belongs.
   
   If specified, the group name must already have been defined.

The use of the WSEGWELL and WSEGEXSS keywords is mutually exclusive for a given segment number of a multisegment well.

Example

```
-- Establish associations between segments and secondary well names
WSEGWELL
-- Well  Segment  Sec   Group
-- Name  Number  Name  Name
    PROD    4   PROD-S4   FIELD /
    INJ    21   GLIFT21 /
/
```

Keywords

2725
**WSEPCOND**  Specify the separator conditions for wells

This keyword associates separator conditions with a well. In compositional runs `WSEPCOND` must be used to assign separator conditions to a well.

The keyword is followed by any number of records, each containing the items of data described below, and each terminated with a slash (/).

The records can be terminated early with a slash, the remaining items taking default values. Defaults can also be specified by a null repeat count of the form \( n^* \), where \( n \) is the number of consecutive items to be defaulted.

The set of records is ended by a blank record, containing only a slash.

1. **Well name, well name template, well list or well list template**
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the *ECLIPSE Technical Description* for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword `WLIST`.

2. **The name of the separator associated with the well.**
   
   This must be a mnemonic previously entered with `SEPCOND`.

**Related keywords**

For well production, surface volumes are calculated using a well separator identified using the `WSEPCOND` keyword. If a well separator is not defined, the group separator defined by the `SEPCOND` keyword is used. If the group separator is not defined, the field separator defined by the `FIELDSEP` keyword is used. If the field separator is not defined, a single flash to standard conditions given by `STCOND` is used.

**Example**

```
WSEPCOND
P11 SEP1 /
'A*' SEP2 /
/
```

---

Keywords

2726
This keyword is used to input the two-phase water relative permeabilities in GASWAT or CO2STORE (non-thermal) cases. The gas relative permeability and the gas-water capillary pressure are entered using the GSF keyword.

The data specified consists of a number of tables of two-phase water saturation data, each terminated with a slash (/). Each table consists of two columns of data. The number of tables should be equal to be number of saturation tables in the run, specified by the TABDIMS keyword, and defaulting to 1.

Column:
1. The water saturation. 
   Values should be between 0 and 1 and should increase monotonically down the column.
2. The corresponding water relative permeability. 
   Values should be between 0 and 1 and should be level or increasing down the column. The first value in the column must be 0.

There must be the same number of entries in each column of a given table, and the number of rows should be less than or equal to the maximum number of rows of saturation data specified with TABDIMS.

Example

<table>
<thead>
<tr>
<th>WSF</th>
</tr>
</thead>
<tbody>
<tr>
<td>0    0</td>
</tr>
<tr>
<td>.2   0</td>
</tr>
<tr>
<td>.4   .004</td>
</tr>
<tr>
<td>.5   .036</td>
</tr>
<tr>
<td>.6   .146</td>
</tr>
<tr>
<td>.7   .42</td>
</tr>
<tr>
<td>.78  1.00 /</td>
</tr>
</tbody>
</table>

Keywords

2727
WSOLVENT  Sets solvent fraction for gas injection wells

This keyword is used to specify the fraction of solvent in the injection stream of each gas injection well, when the solvent model is active (keyword SOLVENT in the RUNSPEC section).

If the WSOLVENT keyword does not appear in the injection schedule, the solvent fraction will be assumed to be zero. Data should only appear for wells which are currently declared to be gas injection wells.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. The fraction of solvent in the injection stream for the well. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

End data record with a slash (/).

End the set of records with a blank record, containing just a slash (/).

Example

```
WSOLVENT
 INJ1  1.0 /
 INJ2  0.0 /
 INJ3  0.5 /
 /
```
WSURFACT  Sets surfactant concentration for injection wells

This keyword is used to specify the concentration of surfactant in the injection stream of each well, when the surfactant flood model is active. If the WSURFACT keyword does not appear in the injection schedule, then concentration values of zero are assumed. Data should only appear for wells that are currently declared to be water injection wells.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. The concentration of surfactant in the injection stream for the well. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

   UNITS: kg/sm\(^3\) (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm\(^3\) (PVT-M)

End data record with a slash (/).

End the set of records with a blank record, containing just a slash (/).

Example

```
WSURFACT
INJS1  0.3 /
INJS2  0.2 /
/ 
```

Keywords

2729
WTADD  Adds to a well operating target or limit

This keyword can be used to add (or subtract) a specified amount to a well control target or limit. The control data must initially have been fully specified using WCONPROD or WCONINJE. This keyword should not be used on a history matching well (see keywords WCONHIST and WCONINJH).

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Definitions of the control or constraint quantity to be changed are listed below. This item can be a user defined quantity (UDA) and can be set with the keywords UDQ and UDADIMS. The values in the brackets are the corresponding values to identify the control quantity if UDA is used for this item.

   ORAT [1]   Oil rate
   GRAT [3]   Gas rate
   LRAT [4]   Liquid rate
   CRAT [5]   Linearly combined rate (coefficients must be set with LINCOM)
   RESV [6]   Reservoir fluid volume rate
   BHP [7]    Bottom hole pressure
   THP [8]    Tubing head pressure (A non-zero VFP table number must first have been set)
   LIFT [9]   Artificial lift quantity (A non-zero VFP table number must first have been set)
   GUID [10]  Guide rate (For group controlled wells)

   The guide rate can only be modified if the well has a user-specified guide rate set for a given phase in keyword WGRUPCON.

   ECLIPSE 300 only

   ECLIPSE 300 only

3. Amount to add for this quantity. This quantity can be a UDA and can be set with the keywords UDQ and UDADIMS.

   UNITS: the units depend upon the quantity chosen. Negative numbers are allowed. The appropriate units are listed in the specification for keywords WCONPROD and WCONINJE.

4. Number of times addition will be performed. Note that addition will be performed at report timesteps only and this item is active only when a UDA is used for item 3. This item will be truncated to the
greatest integer lower than or equal to its value. Also note that any value less than 1 will be defaulted to 1.

DEFAULT: 1

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also keyword WELTARG.

Example

<table>
<thead>
<tr>
<th>WTADD</th>
<th>PROD1</th>
<th>ORAT</th>
<th>200</th>
<th>/</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>PROD2</td>
<td>RESV</td>
<td>1000</td>
<td>/</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>/</td>
</tr>
</tbody>
</table>
WTAKEGAS  Order in which subtractions are taken from gas production

This keyword specifies the order in which fuel, sales and reinjection gas are extracted from the production gas stream of a source group or a reinjection control group. See the notes below for a definition of these groups.

All six possible orderings are allowed: FRS, FSR, RFS, RSF, SFR, SRF.

The default is FSR - fuel (keyword GRUPFUEL), then sales (keyword GRUPSALE), the remainder being available for reinjection. If there is insufficient gas to satisfy all the three requirements, these are prioritized in the order given.

Possible data item values are:

FSR  Fuel, then Reinjection, then Sales
FSR  Fuel, then Sales, then Reinjection
RFS  Reinjection, then Fuel, then Sales
RSF  Reinjection, then Sales, then Fuel
SFR  Sales, then Fuel, then Reinjection
SRF  Sales, then Reinjection, then Fuel.

Notes

- A source group is a group whose vapor production supplies the stream for gas injection. For group injection (see keyword GINJGAS or GRUPINJE), or for well injection (see keyword MINJGAS or WELLINJE), the source group is named in item 3 if the fluid injection stream in item 2 is GV. (For injection streams other than GV, there is no source group.)

- If the injection stream comes from a source group, the amount of gas from this source that is available to reinject is taken into account when calculating injection rates if the availability flag is set (that is, if keyword WAVAILIM has been entered). The gas available to reinject is by default equal to the source group’s gas production rate minus its fuel rate minus its sales target. However, by entering the appropriate item in WTAKEGAS it is possible to prioritize reinjection ahead of fuel and/or sales. Any advance gas allocated to the group (keyword GADVANCE) is also added to the available gas.

- If the injection stream comes from a single separator stage (see GINJGAS item 5), the composition of the fuel and sales gas depends on the amount that is reinjected. When there are fuel or sales targets that take precedence over injection, the injection calculation is still performed first, in order to determine the composition of the gas left for fuel and sales, but the injection rate is limited by the requirement to satisfy the higher priority fuel and sales targets.

- A reinjection control group is a group whose gas production rate is used to control the reinjection rate. In GCONINJE it is the group named in item 11. In GRUPINJE it is the group named in item 12 if item 11 is set to G. In WELLINJE it is the group named in item 14 if item 13 is set to G. A reinjection control group is only relevant if a non-zero reinjection fraction has been set (GCONINJE item 6, GRUPINJE item 10, WELLINJE item 12).
• If a specified fraction of a control group’s surface gas production rate is being reinjected, the reinjection fraction multiplies the control group’s available produced gas rate. The available produced gas rate is by default equal to the control group’s gas production rate, plus any advance gas allocated to it (keyword GADVANCE), minus its fuel rate minus its sales target. However, if R precedes F in the WTAKEGAS item then its fuel rate is not subtracted. Similarly, if R precedes S in the WTAKEGAS item then its sales target is not subtracted.

• The fuel, sales and reinjection gas account calculations are performed at the group level with a reinjection target/limit set in item 6 on GCONINJE. If a group with fuel/sales/reinjection requirements contains gas injection wells that are not subject to group control (see item 2 of WGRUPCON), then these wells may inject gas at a rate that exceeds the reinjection rate calculated for the group after fuel and sales requirements are met (in an FSR case). In this case, the fuel and sales requirements will be equally scaled back so that the total gas usage does not exceed the gas production for the group. If item 95 of OPTIONS3 is set to 1, the lowest priority out of the fuel and sales gas will be scaled back first. If this item in OPTIONS3 is set to 2, no scaling back of gas usage will be performed and it will thus exceed the gas production.

See also "Group injection control options" in the ECLIPSE Technical Description.

Example

```
WTAKEGAS
RSF /
```
WTEMP

**Sets the temperature of an injection well**

This keyword is used to specify the value of the temperature of the injected fluid, when the Temperature option is active (keyword TEMP in the RUNSPEC section).

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.
   
   The well(s) must previously have been declared as injector(s).

2. The injected fluid temperature.
   
   \( \text{UNITES: } ^{\circ}\text{C (METRIC), }^{\circ}\text{F (FIELD), }^{\circ}\text{C (LAB), }^{\circ}\text{C (PVT-M)} \)

End data record with a slash (/).

End the set of records with a blank record, containing just a slash (/).

**Example**

```
WTEMP
INJ1  40.0 /
INJ2  45.0 /
/ 
```
The WTEMPQ keyword writes a list of all the currently defined wells matching the supplied well name or well list template to the PRT file.

The keyword exercises the template matching capabilities of ECLIPSE. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLST.

2. LGR name

   The name of an existing LGR within which the query is to be applied. If the specified LGR is not found the query will be applied to all the wells in the field.

   **ECLIPSE 100**

   For LGR amalgamations, any LGR within the amalgamation may be specified

   DEFAULT: Apply query to all wells in the field

**Example**

```bash
WTEMPQ
   \"*PROD*\" / 
   \"INJ\"  \"LGR-1\" / 
   \"*WLIST?\" / 
/ 
```

---

**Keywords**

ECLIPSE Reference Manual

2735
Instructions for periodic testing of closed wells

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Testing interval

This must be greater than zero.

UNITS: day (METRIC), day (FIELD), hr (LAB), day (PVT-M)

The well is tested at the beginning of the first timestep that starts after the specified interval has elapsed since it was closed. Subsequent tests are performed at each timestep that starts after the specified interval has elapsed since the previous test.

If the testing interval is very small (for example, 1 day), the well is tested at the beginning of each timestep (unless of course the timesteps are smaller than this).

3. Closure reason(s) valid for testing

A string of up to 5 characters.

If the string contains a P: The well is tested if it has been closed for any physical reason (for example failure to operate at its BHP or THP limit). It is opened if the test shows that it can operate again.

If the string contains an E: The well is tested if it has been closed due to an economic limit set in the keywords WECON, CECON or, for ECLIPSE 100, WECONINJ. If it has at least one open connection, the well is opened if it obeys all the limits set in WECON. Any closed connections will not be reopened. However, if the well has no open connections, it must have been closed after a workover resulting from a water cut, GOR or WGR limit violation. In this case, all connections that were closed by automatic workovers are tested individually. A connection is reopened if its water cut, GOR and WGR do not exceed the limits set in WECON and CECON. Connections that were closed manually are not tested. The well is opened if at least one connection is reopened.

If the string contains a G: The well is opened if it has been closed due to a GROUP or FIELD limit set in keyword GECON or the GCON----family. The well is opened unconditionally, since it cannot be tested in isolation. If it causes a group or field limit to be broken, the remedial action is performed at the end of the timestep in the usual manner. If the well has no open connections, all connections closed by automatic workovers are reopened.

ECLIPSE 100 only

If the string contains a D: The well is tested if it has been closed on exceeding its maximum THP design limit set in keyword WTHPMAX. It is reopened if its THP has fallen below a maximum limit for reopening, when flowing at a specified rate (subject to obeying its other rate and pressure limits). The rate limit applied for the test, and the maximum THP for reopening, are set in keyword WTHPMAX.
If the string contains a C, \textbf{while the well is still open}. A connection is reopened if it does not violate any of the limits set in \texttt{WECON} or \texttt{CECON}. Any connections that were closed manually are not reopened. The difference between this type of test and the test triggered by an E is that the E test is only performed after the whole well has been closed by an economic limit, while the C test is performed while one or more connections have been closed but the well is still producing through other connections. The testing interval (item 2) prevents testing until the specified number of days have elapsed since the most recent automatic closure of any connection in that well. Since the well itself does not have to be reopened after a successful C test, the start-up time (item 5) does not apply.

Wells (and, with the C option, connections) that were closed manually are not tested.

\textbf{DEFAULT:} ' ' (an empty string, which turns off the testing process for the well)

4. Number of times the well can be tested
An integer between 0 and 32767
After the well has been tested this number of times (regardless of whether any tests are successful), it is not tested any more unless this item is reset in a subsequent use of the \texttt{WTEST} keyword.
A value of 0 allows the well to be tested an unlimited number of times.
\textbf{DEFAULT:} 0

\textbf{ECLIPSE 100 only} 5. Start-up time
The well’s efficiency factor is steadily ramped up over the specified start-up time when it is reopened after a successful test. During each timestep after reopening, its efficiency factor is multiplied by the quantity

\[ \frac{(T - T_o)}{\text{start-up time}} \]

where

\( T \) is the time at the end of the timestep
\( T_o \) is the time the well was reopened

for as long as \( T - T_o < \text{start-up time} \).

Thus if the start-up time is large compared to the timestep size, the well is brought on gradually, avoiding the timestep convergence difficulties sometimes observed when a high-rate well is suddenly opened.

The start-up time also applies to wells turned on by automatic cycling (see keyword \texttt{WCYCLE}).

\textbf{UNITS:} day (\texttt{METRIC}), day (\texttt{FIELD}), hr (\texttt{LAB}), day (\texttt{PVT-M})

\textbf{DEFAULT:} 0.0, which gives an instantaneous start-up.
End the data record with a slash (/).
End the set of records with a blank record containing just a slash.
Example

WTEST
  'PROD*' 100 PE /
  W1 150 P 10 /
/

Keywords

2738
**WTHPMAX  Maximum THP design limit data**

This keyword controls an option to close a well if its tubing head pressure increases above a specified upper limit, such as the maximum pressure design limit for the wellhead. A high THP could occur if the well’s GOR increases, for example due to gas coning. A closed well can be tested periodically for reopening (if requested with keyword *WTEST*). The well is reopened when its THP falls below the specified reopening limit, for example when the gas cone has subsided to cause a sufficient reduction in GOR.

The well’s operating THP is checked against its maximum design limit at the end of each timestep, when the economic limits are checked (for example keyword *WECON*). If the THP exceeds the upper limit, the well is shut or stopped, depending on the instructions in item 9 of keyword *WELSPECS*. The action takes place at the end of the timestep, so the THP limit is exceeded for one timestep. However, in common with the economic limits, if the limit is violated by proportionately more than the tolerance fraction set in keyword *WLIMTOL* ECLIPSE repeats the timestep after closing the well.

Wells closed on violating their maximum THP limit may be tested periodically to see if they can be reopened. The testing must be requested with keyword *WTEST*, and the closure reason string in item 3 of that keyword must contain a D. During testing, the THP of the well is calculated when the well is flowing either at a specified rate of a given phase, or the rate it was flowing at when it was last closed, subject to obeying its BHP limit and any rate limits on other phases set in keyword *WCONPROD* or *WELTARG*. The test is performed at the beginning of the timestep, using the current conditions in the grid cells containing the well’s connections. If the well’s THP has fallen below a specified upper limit, the well is reopened and flows until closed again by its maximum THP design limit or for any other reason.

The *WTHPMAX* keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See “Well name and well list template matching” in the *ECLIPSE Technical Description* for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword *WLIST*.

2. Maximum THP design limit
   The well will be shut or stopped (see item 9 of keyword *WELSPECS*) at the end of the timestep if its operating THP exceeds this limit.
   A default, zero or negative value switches off this limit.
   **UNITS**: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

3. Rate limit applied (see item 4) when testing the well to see if it can be reopened after being closed on violating its maximum THP limit
   **ORAT** Oil rate
   **WRAT** Water rate
   **GRAT** Gas rate
   **LRAT** Liquid rate

---

**Keywords**

**ECLIPSE Reference Manual**

**2739**
RESV  Reservoir fluid volume rate

DEFAULT:  ORAT

4. Value of the rate limit (see item 3) applied when testing the well to see if it can be reopened
   The units depend upon the quantity chosen in item 3. The appropriate units are listed in the
   specification for keyword WCONPROD.
   A default, zero or negative value sets the value of the rate limit equal to the well’s flow rate of the
   associated quantity when it was last closed after exceeding its maximum THP design limit.

5. Maximum THP limit for reopening the well during a test
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
   The well will be reopened if its THP has fallen below the value set here. The value must not exceed
   the maximum design limit set in item 2.
   A default, zero or negative value applies the same limit as set in item 2.

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Note: items 3, 4 and 5 apply only if the well is to be tested periodically for reopening after being closed on
violating its maximum THP design limit. If you wish to have the well tested periodically, you must set the
appropriate instructions in keyword WTEST, and ensure that the closure reason string in item 3 of that
keyword contains a ‘D’.

Example

<table>
<thead>
<tr>
<th>WTHPMAX</th>
</tr>
</thead>
<tbody>
<tr>
<td>-- well max THP testing testing max THP</td>
</tr>
<tr>
<td>-- name limit limit limit value to reopen</td>
</tr>
<tr>
<td>PROD1 1000 ORAT 500 800 /</td>
</tr>
<tr>
<td>PROD2 1000 RESV 1* 800 /</td>
</tr>
<tr>
<td>/</td>
</tr>
</tbody>
</table>
**WTMULT**

**Multiplies a well operating target or limit**

This keyword can be used to multiply a control target or limit for a well by a specified factor. The control data must initially have been fully specified using WCONPROD or WCONINJE. This keyword should not be used on a history matching well (see keywords WCONHIST and WCONINJH).

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. **Well name, well name template, well list or well list template**
   
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Definitions of the control or constraint quantity to be changed are listed below. This item can be a user defined quantity (UDA) and can be set with the keywords UDQ and UDADIMS. The values in the brackets are the corresponding values to identify the control quantity if UDA is used for this item.

   - ORAT [1] Oil rate
   - LRAT [4] Liquid rate
   - CRAT [5] Linearly combined rate (coefficients must be set with LINCOM)
   - RESV [6] Reservoir fluid volume rate
   - BHP [7] Bottom hole pressure
   - THP [8] Tubing head pressure (A non-zero VFP table number must first have been set)
   - LIFT [9] Artificial lift quantity (A non-zero VFP table number must first have been set)
   - GUID [10] Guide rate (For group controlled wells)
     
     The guide rate can only be modified if the well has a user-specified guide rate set for a given phase in keyword WGRUPCON.

   **ECLIPSE 300 only**

   **ECLIPSE 300 only**

3. **Multiplying factor for this quantity. This quantity can be a UDA and can be set with the keywords UDQ and UDADIMS.**

   UNITS: dimensionless

4. **Number of times multiplication will be performed. Note that multiplication will be performed at report timesteps only and this item is active only when a UDA is used for item 3. This item will be truncated**

---

**Keywords**

2741
to the greatest integer lower than or equal to its value. Also note that any value less than 1 will be defaulted to 1.

DEFAULT: 1

End data record with a slash (/).

End the set of records with a blank record, containing just a slash.

See also keyword **WELTARG**.

### Example

```
WTMULT
    PROD1  ORAT  1.5 /
    PROD2  RESV  0.8 /
```
WTRACER  Sets tracer concentrations for injection wells

This keyword specifies the value of the concentration of a tracer in the injection streams of its associated phase. If a tracer does not appear in the list of tracer names in the WTRACER keyword, a concentration of 0.0 is assumed.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template.
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.
   Each well must previously have been declared as an injector.

2. Name of the tracer
   (Up to 3 characters)

3. Value of the tracer concentration in the injection stream, $T_{conc}$. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

   ECLIPSE 100
   If a cumulative tracer factor is specified in item 4, then $T_{conc}$ is taken as the maximum tracer concentration.

   ECLIPSE 100 only
   4. The value of the cumulative tracer factor, $T_{cum}$. This quantity can be a user defined argument (UDA), and can be set with keywords UDQ and UDADIMS.

   The cumulative tracer factor allows the tracer concentration to be specified as a linear function of the total cumulative injection of the well. This can be useful when modeling $\text{H}_2\text{S}$ generating processes.

   If a cumulative tracer factor is specified, then the tracer concentration in the injection stream is given by:

   \[
   T_c = \min (CI \cdot T_{cum}, T_{conc})
   \]

   Eq. 3.282

   where

   CI   Total cumulative injection at the previous timestep

   $T_{cum}$  Cumulative tracer factor

   $T_{conc}$  Tracer concentration entered in item 3.

   UNITS: 1/sm$^3$ (METRIC), 1/stb for liquid phase tracers, 1/Mscf for gas phase tracers (FIELD), 1/scc (LAB)

   ECLIPSE 100 only
   5. The group whose produced tracer concentration should be used for the tracer concentration of the injection stream for the well.

   If no group is specified, the concentration in item 3 will be used.

   End data record with a slash (/).
End the set of records with a blank record, containing just a slash (/).

Note: When using group reinjection, the concentration used by the well will lag the group’s concentration by one timestep.

Example

```
WTRACER
INJ1  WT1  1.0 /
INJ1  WT2  0.0 /
INJ2  WT1  0.0 /
INJ2  WT2  1.0 /
INJ3  IGS  1.0 /
/
```
WVFPDP Adjusts a well’s BHP obtained from the VFP tables

This keyword specifies a pressure adjustment, which is added to the value of the well BHP obtained by interpolating the VFP tables. It can be used, for example, to help match a well’s flow rate at a given THP, by adjusting the effective pressure loss between the bottom hole and the tubing head. A positive adjustment increases the BHP and makes a production well less productive. Conversely, a negative adjustment improves the production. The reverse holds for injection wells.

In addition, a tubing pressure loss scaling factor may be specified. ECLIPSE adjusts the well’s BHP obtained from the VFP table by multiplying the tubing pressure loss (BHP - THP) by this factor. A scaling factor greater than 1.0 increases the BHP and make a production well less productive.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/). The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. ΔP the pressure adjustment to be added to the well’s BHP
   UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

3. Tubing pressure loss scaling factor, F_p
   The well’s BHP will be adjusted to BHP* according to the equation
   \[
   BHP^* = \text{THP} + F_p \cdot (\text{BHP}_{\text{tab}} - \text{THP})
   \]
   Eq. 3.283
   DEFAULT: 1.0

Note: ECLIPSE automatically adjusts the well’s BHP to take account of any difference between its BHP reference depth (item 5 in keyword WELSPECS) and that of the VFP table (item 2 in record 1 of VFPPROD or VFPINJ), by adding or subtracting a hydrostatic pressure correction based on the density of the fluid in the well bore. This is done independently of the use of this keyword.

Example

```
WVFPDP
  'P*'   10.0 /
  PROD2  -14.3 /
  PROD3   0.0  1.15 /
/
```

Keywords

2745
WVFPEXP  Options to overcome certain problems with THP controlled wells

The WVFPEXP keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template
   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the ECLIPSE Technical Description for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword WLIST.

2. Implicit/explicit flag
   (See Note 1, below)
   IMP The well interpolates its VFP table with the implicit (latest) values of the water and gas flowing fractions
   EXP The well interpolates its VFP table with the explicit (previous timestep’s) values of the water and gas flowing fractions
   DEFAULT: IMP

3. Close the well if it is found to be operating on the stabilized part of its VFP curve?
   (See Note 2, below)
   NO The check is not performed.
   YES At the end of each timestep, ECLIPSE checks the well if it is a THP-controlled producer to see if it is operating on a stabilized region of its VFP curve, identified by a zero gradient of BHP versus flow rate. If the well is found to be operating in this region it is closed.
   DEFAULT: NO

4. Prevent the well changing from rate control to THP control when it is constrained to operate on the unstable side of its VFP curve?
   (See Note 3, below)
   NO In ECLIPSE 100, changes in the control mode are not prevented. Under some circumstances the well may ‘hunt’ between THP control and rate control.
   YES1 While operating on rate control (including group control), the well is not switched to THP control if it is constrained by a low rate limit to operate on the unstable side of the VFP curve, provided that the well would produce at a higher rate under THP control.

Note: In ECLIPSE 300 the switch to THP control is always prevented if this would result in hunting. This item only controls whether or not this action is reported.
A message is printed the first time the well is prevented from changing its control mode.

YES2 As YES1, except that a message is printed every time the well is prevented from changing its control mode.

DEFAULT: NO

5. Control on how VFP tables are extrapolated for water fraction, gas fraction and Artificial Lift Quantity (ALQ) values that fall outside the range of values in the table.

VFP tables for production wells are constructed with a range of values for flow, THP, water fraction, gas fraction and ALQ. If the table is queried with a value that falls outside of its table range, then in the case of flow and THP, the table is linearly extrapolated in this value. For water fraction and gas fraction values, linear extrapolation can result in non-monotonic or non-U shaped VFP curves; such curves may cause convergence problems, or the well to die prematurely. This item allows you to restrict the extrapolation in water and gas fractions and, for back compatibility, ALQ values.

WG Constant value extrapolation is used for water fraction and gas fraction values in VFP table look ups. So, for example, if a VFP table is queried with a gas fraction that is greater than the maximum value in the table, the look up will be made at the table’s maximum gas fraction. The table is linearly extrapolated in ALQ.

WGA Constant value extrapolation is used for water fraction, gas fraction and ALQ values in VFP table look ups. Prior to the 2005A release, this was the extrapolation behavior of ECLIPSE 300.

NONE Linear extrapolation is used for water fraction, gas fraction and ALQ values that fall outside of the bounding values in the VFP table. This is the ECLIPSE 100 behavior.

DEFAULT: WG

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.

Notes

Implicit/explicit water and gas fractions

When a well is under THP control, its flow rate is determined by the intersection of its inflow performance relationship (IPR) and the VFP curve interpolated at the current value of its water and gas flowing fractions, on a plane of BHP versus flow rate. Often the VFP curve is U-shaped, and when there are two intersections the one with the higher flow rate is chosen (the other being unstable). If the IPR does not intersect the VFP curve, the well is deemed to have died.

Engineering calculations of the intersection point usually plot the VFP curve at a constant water cut and GOR. However, if the well is completed in a number of layers having different water cut values, the overall water cut of the well will depend on the BHP, especially if due to poor vertical communication the layers are at different pressures. Because ECLIPSE uses the latest (implicit) values of the water and gas fractions to interpolate the VFP tables, it sees a VFP curve that is plotted at water cut and GOR values that vary with the BHP. This can alter the shape of the VFP curve away from its usual monotonic or U-shaped nature, perhaps introducing extra minima and unstable regions. If the well enters one of these extra unstable regions, ECLIPSE thinks the well has died and will close it prematurely.
This problem can be overcome by using the explicit (previous timestep’s) values of the well’s flowing water and gas fractions to look up the VFP table. The VFP curve is no longer be distorted by the variation of water cut or GOR with BHP, thus reducing the risk of premature shut-in. The \texttt{WVFPEXP} keyword can be used to designate particular wells to have their VFP tables interpolated at explicit values of the water and gas fractions. Furthermore, if ECLIPSE detects that a well has entered an unstable region of the VFP curve while using implicit values of the flowing fractions, it will automatically change the well to use explicit values, and print a warning message.

\textbf{Note:} Explicit water and gas fractions are only used to look up the VFP tables to obtain the flowing BHP. The actual water cut and GOR of the well are still calculated implicitly as usual.

\textbf{ECLIPSE 100}

Item 2 essentially replaces \texttt{OPTIONS} keyword item 9. However, for back-compatibility, if \texttt{OPTIONS} item 9 is set greater than zero then all wells use explicit values of the water and gas fractions to interpolate the VFP tables, irrespective of any instructions in \texttt{WVFPEXP}.

\section*{Stabilized VFP curves}

Another use of the \texttt{WVFPEXP} keyword is to activate a check for wells operating on the flat part of a stabilized VFP curve. The VFP \texttt{i} program can output stabilized VFP tables, in which the unstable part of the VFP curves to the left of the minimum is replaced with a horizontal line at the minimum BHP value. This can overcome problems caused by the sudden death of a well when its IPR drops off the minimum of the VFP curve, since the flow continuously reduces to zero as the grid block pressure decreases instead of stopping suddenly. However, it does allow wells to continue to operate at lower flow rates when physically they would be unable to flow. By setting item 3 in this keyword, ECLIPSE can be instructed to shut the well in at the end of the timestep when it has converged on a solution in the stabilized part of the VFP curve, which is identified by a zero gradient of BHP versus flow rate. In ECLIPSE 100, the switch also prevents wells being reopened by the \texttt{WTEST} keyword if they would be operating on the flat stabilized part of the VFP curve.

\section*{Flow limits within the unstable region}

The U-shape of a VFP curve can also cause problems if a well is constrained by a rate limit (or a group control rate target) to operate within the unstable part of the curve. In this region the THP of the flowing well decreases as the flow rate decreases, and at a low enough flow rate the THP may fall below its minimum limit. The well would normally then switch to THP control, and it could either start hunting between control modes or die prematurely. ECLIPSE 100 and ECLIPSE 300 differ in their behavior at this point as described below.

\textbf{ECLIPSE 100}

By setting item 4 to \texttt{YES1} or \texttt{YES2} the well is prevented from changing to THP control as long as it would produce at a \textbf{higher} rate under THP control. Under these circumstances the well continues to operate at its rate constraint, while its calculated THP is allowed to fall below the minimum limit. But it is still allowed to die if its IPR falls off the bottom of the VFP curve. This item replaces \texttt{item 73} in \texttt{OPTIONS} keyword, which was formerly used to prevent control mode switching from happening in these circumstances for all wells.

\textbf{ECLIPSE 300}

The well will \textbf{always} be prevented from changing to THP control if it would produce at a \textbf{higher} rate under THP control. Item 4 only controls the reporting of this action.

\section*{Example}

\begin{verbatim}
WVFPEXP
 'PROD*'  EXP  YES  YES1 /
\end{verbatim}
Individual well block average pressure controls

This keyword controls the calculation of well block average pressures for individual wells. The well block average pressures are computed using the same formulae as those employed by the \texttt{WPAVE} keyword. They represent the average pressure of the grid blocks containing connections to a given well, and optionally their adjacent and diagonal neighbors. Neighbor contributions can be weighted according to either the connection transmissibility factors or the grid block pore volumes. The \texttt{WWPAVE} keyword overwrites any current controls specified by the \texttt{WPAVE} keyword and in turn can be overwritten by any subsequent use of the \texttt{WPAVE} keyword.

The average pressures can be written to the Summary file using the \texttt{SUMMARY} section keywords \texttt{WBP}, \texttt{WBP4}, \texttt{WBP5} and \texttt{WBP9}. Well PIs based on these average pressures can also be written to the Summary file using the \texttt{SUMMARY} section keywords \texttt{WPI1}, \texttt{WPI4}, \texttt{WPI5} and \texttt{WPI9} (see "SUMMARY section").

The averages relate to a Cartesian grid and should not be used in a radial grid. The averages are used for reporting purposes only, and will not affect any other results.

The keyword is followed by any number of records, each containing the following items of data, and each terminated with a slash (/).

The set of records must end with a blank record, containing only a slash (/).

1. Well name, well name template, well list or well list template

   A template enclosed in quotes can be used to refer to multiple wells or well lists. See "Well name and well list template matching" in the \textit{ECLIPSE Technical Description} for further details. Well list names should be enclosed in quotes and begin with an asterisk (*). Well lists are constructed with the keyword \texttt{WLIST}.

2. F1, the weighting factor between the inner block and the outer ring of neighboring blocks, in the connection factor weighted average.

   If the value lies between 0.0 and 1.0, the average is calculated by equation 3.236. The value 1.0 gives total weighting to the inner blocks, containing the well connections. The value 0.0 gives total weighting to the 4 or 8 blocks neighboring each inner block.

   A value of $F_1 < 0.0$ is used to indicate that the pressure of the inner block and its outer ring of neighboring blocks should be averaged according to their pore volumes. The average block pressure for each connection is calculated by equation 3.237.

   DEFAULT: 0.5

   \textbf{Note:} For one-block averages (\texttt{WBP}) $F_1$ is effectively 1.0, whatever the value entered here.

3. F2, the weighting factor between the connection factor weighted average and the pore volume weighted average, used by 3.234.

   The value should lie between 0.0 and 1.0. The value 1.0 gives a purely connection factor weighted average, and 0.0 gives a purely pore volume weighted average.

   DEFAULT: 1.0

4. Depth correction flag

   This flag controls how the grid block pressures are corrected to the well’s bottom hole reference depth (see item 5 in keyword \texttt{WELSPECS}), or to an alternative reference depth if specified in keyword
WPAVEDEP, by applying a hydrostatic head between each grid block center and the well’s reference depth.

WELL The hydrostatic head is calculated using the density of the fluid in the wellbore at the well connections.

RES The hydrostatic head is calculated using a representative density for the fluid in the reservoir. This density is calculated by averaging over fluid density for all the grid blocks associated with the well’s connections. The average over phases is weighted by the phase saturation, and the average over grid blocks is weighted by pore volume. Whether the averaging is performed over all grid blocks with declared connections to the well or only those with currently open connections is determined by item 4.

NONE Grid block pressures are not depth corrected.

DEFAULT: WELL

Note: The wellbore fluid density is set to zero whenever the well is shut. Thus if WELL is selected there is a discontinuity in the reported pressure average when the well’s status changes between shut and open/stopped.

5. Well connection flag

This flag controls whether the grid blocks associated with all the well’s declared connections contribute to the average pressure, or just those associated with the currently open connections.

OPEN Only grid blocks associated with currently open connections are included in the averaging calculation.

ALL Grid blocks associated with all currently defined connections (whether open or closed) are included in the averaging calculation.

DEFAULT: OPEN

Note: If OPEN is selected there is a discontinuity in the reported pressure average whenever new connections are opened or existing ones are closed. This may be avoided by selecting ALL and defining all the well’s connections at the start of the run (whether initially open or closed).

If this keyword is not present, all items assume their default values, giving a connection factor weighted average, evenly weighted between the inner blocks and the outer ring of neighbors. The depth correction uses the wellbore density and only grid blocks associated with currently open well connections are included in the average.

Note: In dual porosity runs the wells are completed in the fracture system. A WBPn average reports the average pressure in the surrounding fracture cells.

End the data record with a slash (/).

End the set of records with a blank record, containing just a slash.
Example

WWPAVE
  PROD1  1*   0.0   1*   ALL /
  PROD2  1*   0.0   1*   ALL /
/

Keywords

2751
This section contains an alphabetic listing of keywords starting with the letter X.
Specifications cell initial oil composition

This keyword specifies the initial oil composition values for each cell. The keyword should be followed by \( N_x \cdot N_y \cdot N_z \cdot N_c \) values in normal order, with the X index changing fastest. For thermal runs with water \( N_x \cdot N_y \cdot N_z \) extra values are required, all set to zero.

This keyword may be used to specify the initial solution explicitly, as an alternative to equilibration using \texttt{EQUIL}, the usual method of initializing a compositional run.

Explicit initialization is expected if the \texttt{PRESSURE} keyword is present. The same information may be obtained from a Restart file using the \texttt{GETSOL} keyword, or composition may be entered as a function of pressure using \texttt{XMVF}.

When using explicit initialization, you must ensure that the initial solution is stable and physically reasonable. This is potentially difficult in the case of two-phase hydrocarbon cells, and \texttt{XMF} should be used with caution.

\texttt{XMF} values that do not sum to 1.0 are normalized provided the error is less than 10%. An error greater than 10% is flagged as a data error.

See "Initial composition with respect to depth" in the \textit{ECLIPSE Technical Description} for further information.

Example

\begin{verbatim}
XMF
300*0.0021
300*0.0294
300*0.6599
300*0.0869
300*0.0591
300*0.0967
300*0.0472
300*0.0153
300*0.0034 /
\end{verbatim}
XMFVP

Liquid composition with respect to pressure tables

The data comprises a table of liquid composition with respect to pressure data for each equilibration region. Each table consists of $N_c$ columns of data, and is terminated with a slash (/).

Column:

1. Pressure values.
   The values should increase monotonically down the column.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2 to $N_c + 1$. The corresponding values of $x_i$, the liquid phase mole fractions.
   The $N_c$ mole fractions specified must add up to unity, and the program checks that this is the case.
   For pressure values outside the interval specified, constant value extrapolation is performed.

The default maximum number of rows per table is 50; this may be reset using the third item in keyword EQLDIMS. The number of equilibration regions is entered using the first item in keyword EQLDIMS.

See keyword XMF and "Initial composition with respect to depth" in the ECLIPSE Technical Description for further information.

Example

In this case a constant composition with respect to pressure.

```
XMFVP
1000.0 0.0121 0.0194 0.6597 0.0869 0.0591 0.0967 0.0472 0.0153 0.0034
10000.0 0.0121 0.0194 0.6597 0.0869 0.0591 0.0967 0.0472 0.0153 0.0034
/```

XSTC

Specifies oil composition at standard conditions

This keyword specifies the oil composition associated with surface volume rates in the Flux Boundary option.

This is required when a Flux file generated from a black oil run is used in either a compositional or a thermal run.

For further information see "Flux boundary conditions" in the ECLIPSE Technical Description.

Example

```
XSTC .0121 .0194 .6599 .0869 .0591 .0967 .04745 .01515 .0033 /
```
Y

This section contains an alphabetic listing of keywords starting with the letter Y.
Geomechanics region numbers

This keyword is only valid if GEOMECH is specified in the RUNSPEC section. The YLDNUM keyword should be followed by one integer for every grid block in the current input box specifying the region to which parameters set with the keywords GEOYLD, PERMSTAB or BIOTCTAB belong. The region number should not be less than 0 or greater than the GEODIMS item 1.

Grid blocks are ordered with the X axis index cycling fastest, followed by the Y and Z axis indices. Repeat counts may be used for repeated values (for example, 115*2). Spaces must not be inserted on either side of the asterisk.

Example

```
YLDNUM
144*1 96*2 /
```
Specifies cell initial gas composition

This keyword specifies the initial gas composition values for each cell. The keyword should be followed by \( N_x \cdot N_y \cdot N_z \cdot N_c \) values in normal order, with the X index changing fastest. For thermal runs with water, \( N_x \cdot N_y \cdot N_z \) extra values are required, giving the fraction of water in the gas phase.

This keyword may be used to specify the initial solution explicitly, as an alternative to equilibration using EQUIL, which is the normal method of initializing a compositional run.

Explicit initialization is expected if the PRESSURE keyword is present.

The same information may be obtained from a Restart file using the GETSOL keyword, or composition may be entered as a function of pressure using YMFVP.

When using explicit initialization, you must ensure that the initial solution is stable and physically reasonable. This is potentially difficult in the case of two-phase hydrocarbon cells, and YMF should be used with caution.

YMF values that do not sum to 1.0 are normalized provided the error is less than 10%. An error greater than 10% is flagged as a data error.

See "Initial composition with respect to depth" in the ECLIPSE Technical Description for further information.

Example

```
YMF
300*1.0
300*0.0
300*0.0
300*0.0
300*0.0
300*0.0
300*0.0
300*0.0
300*0.0
300*0.0 /
```
YMFVP  

Vapor composition with respect to pressure tables

The data comprises a table of vapor composition with respect to pressure data for each equilibration region. Each table consists of $N_c$ columns of data, and is terminated with a slash (/).

Column:

1  Pressure values.
   The values should increase monotonically down the column.
   UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2 to $N_c + 1$  The corresponding values of $y_i$, the vapor phase mole fractions.
   The NCOMP S mole fractions specified must add up to unity, and the program will check that
   this is the case.
   For pressure values outside the interval specified, constant value extrapolation is performed.

The default maximum number of rows per table is 50; this may be reset using the third item in keyword EQLDIMS. The number of equilibration regions is entered using the first item in keyword EQLDIMS.

See keyword YMF and "Initial composition with respect to depth" in the ECLIPSE Technical Description for further information.

Example

In this case a constant composition with respect to pressure.

```
YMFVP
1000.0  0.0121 0.0194 0.6597 0.0869 0.0591 0.0967 0.0472 0.0153 0.0034
10000.0 0.0121 0.0194 0.6597 0.0869 0.0591 0.0967 0.0472 0.0153 0.0034
/```

Keywords

ECLIPSE Reference Manual

2759
YOUNGMOD  Young’s modulus for rock stress balance

This keyword has no effect if GEOMECH is not specified in the RUNSPEC section.

The keyword should be followed by one non-negative real number for every grid block in the current input box, specifying the rock Young’s modulus. The data must be terminated by a slash (/).

UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

Grid blocks are ordered with the X-axis index cycling fastest, followed by the Y- and Z-axis indices. Repeat counts may be used for repeated values (for example 115*208.4). Spaces may not be inserted on either side of the asterisk.

Note: There is no default for the Young’s modulus. The simulation will stop if the GEOMECH keyword is specified but this data is not supplied.

Example

```plaintext
-- Field units (psi)
YOUNGMOD
  81*1.45E6 81*1.E4 81*1.E5 81*1.8E5 /
```
YSTC

Specifies gas composition at standard conditions

This keyword specifies the gas composition associated with surface volume rates in the Flux Boundary option.

This is required when a Flux file generated from a black oil run is used in either a compositional or a thermal run.

For further information, see "Flux boundary conditions" in the ECLIPSE Technical Description.

Example

YSTC

.0121 .0194 .6599 .0869 .0591 .0967 .04745 .01515 .0033 /
Z

This section contains an alphabetic listing of keywords starting with the letter Z.
**ZCORN**

### Depths of grid block corners

Each grid block has 8 corners. This keyword enables the depths of each corner of each grid block to be separately specified. It is used for specifying depths for corner point geometry.

The ZCORN data are entered for the current box. If a BOX keyword is currently active (ENDBOX has not cancelled it) when ZCORN is encountered, then the number of values entered must be the number for that box. If no BOX keyword is currently active, then the default box spanning the global grid is assumed, which is defined by the DIMENS keyword.

The keyword must be followed by

\[2 \times \text{NDX} \times 2 \times \text{NDY} \times 2 \times \text{NDZ}\]

values, where NDX, NDY, NDZ are the dimensions of the current box. For cell \(i\), the 8 ZCORN values are \(z_{i,1}\), \(z_{i,2}\), \(z_{i,3}\), \(z_{i,4}\), \(z_{i,5}\), \(z_{i,6}\), \(z_{i,7}\), \(z_{i,8}\). Here values 1-4 are for the top face, with \(z_{i,1}\) on the near left corner, \(z_{i,2}\) on the near right corner, \(z_{i,3}\) on the far left corner, and \(z_{i,4}\) on the far right corner. Values 5-8 have the same function for the bottom face. Then the arrangement of the ZCORN values within this range are:

- for the first row of \(\text{NDX}\) cells, input the near top values \(z_{1,1}\), \(z_{1,2}\), \(z_{2,1}\), \(z_{2,2}\),..., \(z_{i,1}\), \(z_{i,2}\),... \(z_{\text{NDX},1}\), \(z_{\text{NDX},2}\);
  followed by the far top values \(z_{1,3}\), \(z_{1,4}\), \(z_{2,3}\), \(z_{2,4}\),..., \(z_{i,3}\), \(z_{i,4}\),... \(z_{\text{NDX},3}\), \(z_{\text{NDX},4}\).

- Repeat for each subsequent row of \(\text{NDX}\) cells in the top plane.

- Now repeat the last two steps for the **bottom** values of the top plane.

- Finally, repeat all previous steps for each plane in the grid.

The resulting data will lie on the grid constructed by the corner points in space, with “touching” corners of adjacent cells separated by a small margin. The separation is only for visualizing the grid, and does not affect the ZCORN values.

The last value is followed by a slash (/).

As the data for this keyword can be quite voluminous, it is suggested that the data should be prepared using a grid pre-processor (for example PETREL).

**UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

**DEFAULT:** <undefined>

If fine control is required in positioning the corner points in local grid refinements (see "Local grid refinement and coarsening" in the ECLIPSE Technical Description), then ZCORN can also be used in a CARFIN or REFIN block. In this case, the ZCORN values will be checked for consistency with the parent grid. The current box here is for the local grid.

If the average value of \(\text{DZ} \leq 10^{-6}\) for any cell, the cell will be either be made inactive or in thermal cases it may become rock-filled.

### Example

Here NDX=3, NDY=2, NDZ=2; and the two z-planes are horizontal with all cells touching exactly.

```plaintext
ZCORN
24*1000
24*1020
24*1020
24*1040 /
```

---

**Keywords**

2763
Critical Z-factors

In a run with $N_c$ components, this keyword associates a critical Z-factor with each component.

These Z-factors are used to determine critical molar volumes using the relationship $V_c = Z_c RT_c / P_c$.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in “Equations of state regions” in the ECLIPSE Technical Description.

DEFAULT: There are no component-wise default values. Users are recommended not to default these to the internal default value.

Example

```
ZCRIT 1.0 0.96 0.994 0.991 /
```
**Critical Z-factors for detailed components**

In a lumped simulation run with \( N_d \) detailed components, this keyword associates a critical Z-factor with each detailed component. Lumped simulations are enabled with the \texttt{LUMPdims} keyword in the \texttt{RUNSPEC} section; further details can be found in "Delumping and lumping" in the \textit{ECLIPSE Technical Description}.

The keyword should be followed by a record of \( N_d \) values, terminated with a slash, for each reservoir equation of state region. The values represent the critical Z-factor of each detailed component, in the order in which the detailed components are defined in the \texttt{LUMPING} keyword.

These Z-factors are used to determine critical molar volumes using the relationship \( V_c = Z_c R T_c / P_c \).

If multiple equations of state are being used, a record of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword \texttt{TABDIMS}. The use of multiple EoS regions is discussed in "Equations of state regions" in the \textit{ECLIPSE Technical Description}.

**Example**

```
ZCRITDET
1.0 0.96 0.994 0.991 /
```
Critical Z-factors for surface EoS

This optional keyword supplies critical Z-factor values for use in the surface calculations. The default, if this is not supplied, is for these to follow the reservoir values supplied with keyword ZCRIT.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

Example

```
ZCRITS
1.0 0.96 0.994 0.991 /
```
ZCRITSDE  Critical Z-factors for surface EoS for detailed components

This optional keyword supplies critical Z-factor values for use in the surface calculations for detailed components in a lumped simulation run. The default, if this is not supplied, is for these to follow the reservoir values supplied with keyword ZCRITDET. Lumped simulations are enabled with the LUMPDIMS keyword in the RUNSPEC section; further details can be found in "Delumping and lumping" in the ECLIPSE Technical Description.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

Example

```
ZCRITSDE
1.0 0.96 0.994 0.991 /
```
ZCRITVIS

Critical Z-factors for viscosity calculations

In a run with $N_c$ components, this keyword associates a critical Z-factor with each component, to be used for viscosity calculations only.

These Z-factors are used to determine critical volumes for use in the Lorentz-Bray-Clark viscosity correlation, using the relationship $V_c = Z_c R T_c / P_c$.

If VCRITVIS or ZCRITVIS is not entered, then values entered with VCRIT or ZCRIT will be used. The form of ZCRITVIS is the same as that of ZCRIT.

If multiple equations of state are being used, a set of data should be entered for each reservoir EoS. The number of reservoir equation of state regions is entered as item 9 of keyword TABDIMS. The use of multiple EoS regions is discussed in "Equations of state regions" in the ECLIPSE Technical Description.

Example

```
ZCRITVIS
1.0 0.96 0.994 0.991 /
```
**ZFACTOR1**

**Z-factor first coefficient**

This keyword can only be used with the **THERMAL** option.

Z-factors are used to determine molar volumes using the relationship \( V = ZRT / P \). In a live oil thermal run the Z-factors are assumed to be:

\[
Z = Z_0 - Z_1 \cdot \frac{P}{RT}
\]

where the coefficient \( Z_0 \) and \( Z_1 \) are given by keywords **ZFACTOR** and **ZFACT1** respectively. They are also used in determining the pressure dependence of the gas component enthalpy

\[
\frac{\partial H}{\partial P} = -Z_1
\]

The coefficient \( Z_1 \) can be determined from the Joule-Thomson coefficient \( \mu^c \) and the average gas molecular weight \( MW_{gas} \) and the average gas specific heat \( C_{gas} \):

\[
Z_1^c = \mu^c \cdot MW_{gas} \cdot C_{gas}
\]

There should be one value for each component.

**UNITS:** m\(^3\)/kg-M (**METRIC**), ft\(^3\)/lb-M (**FIELD**), cc/gm-M (**LAB**), m\(^3\)/kg-M (**PVT-M**)

**DEFAULT:** 0.0

If multiple equations of state are being used, a set of data should be entered for each reservoir. The number of reservoir equation of state regions is entered as item 9 of **TABDIMS**. Use of multiple EoS regions is discussed in "Reservoir equation of state" in the **ECLIPSE Technical Description**.

**Example**

```
ZFACTOR
  1.0 1.0 1.0 1.0 /
ZFACT1
  0.1 0.1 0.1 0.1 /
```
ZFACT1S

Z-factor first coefficient for surface conditions

This optional keyword, together with keyword ZFACTORS can be used with the THERMAL option to modify the surface values of the component Z-factor coefficient. The default values of the surface Z-factor coefficients are the reservoir Z-factor coefficients, defined by keywords ZFACTOR and ZFACT1.

The DENSITY keyword is normally used to determine the surface gas density, so this keyword will have little effect. However, if item 260 of the OPTIONS3 keyword is set to 1, then the surface gas density will be determined from the surface flash results, and data provided by this keyword will have more impact.

This keyword is used in the same way as the ZFACT1 keyword.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of TABDIMS. Use of multiple EoS regions is discussed in "Reservoir equation of state" in the ECLIPSE Technical Description.

Example

```
ZFACTORS
  1.0 0.96 0.994 0.991 /
  
ZFACT1S
  0.0 0.0 0.0 0.0 /
```
ZFACTOR

Z-factor zeroth coefficient

This keyword can only be used with the THERMAL option.

Z-factors are used to determine molar volumes using the relationship \( V = \frac{ZRT}{P} \). In a live oil thermal run the Z-factors are assumed to be:

\[
Z = Z_0 Z_1 \cdot \frac{P}{RT}
\]

where the coefficient \( Z_0 \) and \( Z_1 \) are given by keywords ZFACTOR and ZFACT1 respectively.

There should be one value for each component.

DEFAULT: 0.96

If multiple equations of state are being used, a set of data should be entered for each reservoir. The number of reservoir equation of state regions is entered item 9 of TABDIMS. Use of multiple EoS regions is discussed in "Reservoir equation of state" in the ECLIPSE Technical Description.

Example

ZFACTOR
1.0 1.0 1.0 1.0 /
ZFACT1
0.1 0.1 0.1 0.1 /
ZFACTORZ

Z-factor zeroth coefficient for surface conditions

This optional keyword, together with keyword ZFACT1S can be used with the THERMAL option to modify the surface values of the component Z-factor coefficient. The default values of the surface Z-factor coefficients are the reservoir Z-factor coefficients, defined by keywords ZFACTOR and ZFACT1.

The DENSITY keyword is normally used to determine the surface gas density, so this keyword will have little effect. However, if item 260 of the OPTIONS3 keyword is set to 1, then the surface gas density will be determined from the surface flash results, and data provided by this keyword will have more impact.

This keyword is used in the same way as the ZFACTOR keyword.

If multiple equations of state are being used, a set of data should be entered for each surface EoS. The number of surface equation of state regions is entered as item 10 of TABDIMS. Use of multiple EoS regions is discussed in "Reservoir equation of state" in the ECLIPSE Technical Description.

Example

```
ZFACTORZ
1.0 0.96 0.994 0.991 /
```

```
ZFACT1S
0.0 0.0 0.0 0.0 /
```
Enter overall composition

The data consists of a single composition for each reservoir equation of state region.

Example

```
ZI
0.0121 0.0194 0.6597 0.0869 0.0591 0.0967 0.0472 0.0153 0.0034
```

---

**Keywords**

2773
ZIPP2OFF

**Turns off automatic timestep selection control**

This keyword turns off the automatic timestep selection turned on with the `ZIPPY2` keyword. There is no associated data.
**ZIPPY2**  

**Turns on automatic timestep selection control**

ECLIPSE chooses its timestep by referring to a number of conditions; it takes the most restrictive. These conditions are explained elsewhere. The basic principle is that, subject to accuracy, ECLIPSE takes the largest timestep possible. However, the biggest timestep is not necessarily the most efficient in terms of computational cost per simulated day. Use of this keyword adds another constraint on the timestep selection that takes a smaller timestep if it is predicted that this is more efficient.

The aim of this keyword is to provide a near optimal performance run with one keyword call using no, or minimal, *a priori* knowledge of the problem. This is to be compared with the number of calls to **TUNING** and the amount of *a posteriori* knowledge that would be required to achieve the same result using conventional means.

When using LGRs the timestep restriction applies to the global grid only. This restriction does not apply when used with LGRs-in-place; see the **LGRLOCK** keyword.

The feature can be turned off by using the **ZIPP2OFF** keyword.

---

**CAUTION:** The problem of timestep selection has not been solved for the general case. Using this keyword on many ‘well-behaved’ datasets increases the run time unless the right combination of options is selected—which is no improvement over the **TUNING** keyword. Use of **ZIPPY2** in such cases is not recommended.

There is one group of problems, though, where use of this keyword will make a significant impact on the run times. These are runs that are very choppy, where the “NON-LINEAR EQUATION CONVERGENCE FAILURE” message is occurs throughout the run, or large portions of it.

There follows a full list of possible options within **ZIPPY2**. After this listing a selection of recommended choices is given. All the options must be enclosed in quotation marks ‘’.

Many of the options contain integer or real number data items and these should be input in the usual ECLIPSE manner of ‘**KEYWORD** = integer’ . In some case two numbers are required; these should be separated with a comma, for example, ‘**KEYWORD** =1,5’.

**ECLIPSE 100 only**

- **NEWTON = integer**
  
  Sets the maximum number of Newton iterations.
  
  Included as an alternative to setting it in the **TUNING** keyword. Note that setting it here overwrites any previous value set with the **TUNING** keyword and that any subsequent use of the **TUNING** keyword overwrites any value entered here.

**ECLIPSE 100 only**

- **MINCHOP = real**
  
  Sets the minimum choppable timestep.
  
  Included as an alternative to setting it in the **TUNING** keyword. Note that setting it here overwrites any value set previously with the **TUNING** keyword and that any subsequent use of the **TUNING** keyword overwrites any value entered here.

**ECLIPSE 100 only**

- **MINSTEP = real**

  Sets the minimum timestep
  
  Included as an alternative to setting it in the **TUNING** keyword. Note that setting it here overwrites any value set previously with the **TUNING** keyword and that any subsequent use of the **TUNING** keyword overwrites any value entered here.
ECLIPSE 100 only ' MAXSTEP = real ' Sets the maximum timestep.

Included as an alternative to setting it in the TUNING keyword. Note that setting it here overwrites any value previously set with the TUNING keyword and that any subsequent use of the TUNING keyword overwrites any value entered here.

ECLIPSE 100 only ' NEXTSTEP = real ' Sets the next timestep.

Included as an alternative to setting it in the TUNING keyword. Note that setting it here overwrites any value set previously with the TUNING keyword and that any subsequent use of the TUNING keyword overwrites any value entered here. The same applies to the use of the NEXTSTEP keyword.

ECLIPSE 100 only ' TIGHTEN = real ' Sets the amount to tighten convergence tolerances by.

Included as an alternative to setting it in the TUNING keyword. Note that setting it here overwrites any values set previously with the TUNING keyword and that any subsequent use of the TUNING keyword overwrites any value entered here.

' SIM = real ' Simple timestep selection algorithm

Attempts to select timesteps so that the user-given number of Newtons is needed to solve it. A real number is preferable to an integer. No other keywords are needed if this option is selected.

ECLIPSE 100 only ' AGGRESSIVE=YES ' Can only be used with the SIM option above

' AGGRESSIVE=NO ' Attempts to increase timesteps faster when it feels safe to do so.

This option performs better with higher values of SIM (see example below), and will result in more chops but will also often lead to a reduction in CPU time.

Data

These options determine the amount of data to be used in approximating the cost function. These options can only be used with ECLIPSE 100.

ECLIPSE 100 only Use of ' NPTS ' is compulsory

ECLIPSE 100 only Use of ' PARTITION ' is optional.

ECLIPSE 100 only ' NPTS = integer ' Number of different timesteps for which data is stored.

ECLIPSE 100 only ' PARTITION = integer ' Number of data partitions.

Partitions the data into the specified number of groups by size of timestep. New data overwrites the oldest data in the group to which it belongs. This can stop or delay the loss of data for large timesteps because of being overwritten by newer small timestep data and vice versa.

DEFAULT: 1 Definition of the cost
These options define the cost function (measure of efficiency of a timestep) to be used. These options can only be used with ECLIPSE 100.

**ECLIPSE 100 only**

Use of either ‘CPU’ or ‘SOLVER’ is compulsory.

**ECLIPSE 100 only**

Use of SPRING is optional.

‘CPU’ Uses CPU times from the machine.

---

**CAUTION:** This is the true cost of a timestep and in many ways must therefore be preferred. However, the CPU time ‘flutters’, which then feeds into the timestep selection. If this cost function is used, results are not repeatable, because different timesteps will be used in each run.

**ECLIPSE 100 only**

‘SOLVER = integer1, integer2’

Uses Cost = integer1*number of newtons + integer2*number of linears.

The CPU cost, apart from very small problems, is dominated by the time spent calculating the Jacobian terms for the Newton iteration and solving the resulting linear system. Using this cost function does mean that the same timesteping can be repeated between two runs. Also note that the integer values do not have to be chosen so that cost equals CPU cost; it only needs to be proportional.

**ECLIPSE 100 only**

‘SPRING = real, integer’

Multiplies the cost function by an additional, newton-weighted, factor.

The cost of the timestep, as defined by one of the means above, is multiplied by the following function:

\[
\left( 1 + \text{real} \left( \frac{\text{NEWT}}{\text{NEWTMX}} \right)^{\text{integer}} \right)
\]

where NEWT is the number of Newtons used for this timestep and NEWTMX is the maximum number of Newtons allowed. This non-linearly penalizes timesteps that use more Newton iterations. The situations when this might be useful are described below.

### Fitting function type

These keywords define how the cost function, defined above, is going to be approximated based on the discrete data (defined by data keywords) that has been stored about it. These keywords can only be used with ECLIPSE 100

Use of one of the following is compulsory.

‘BEZIER = integer’

Uses a Bezier polynomial with integer coefficients to fit the data.

Note that integer coefficients means that the resulting polynomial is of degree integer-1.

‘BEZIER2 = integer’

Uses a Bezier polynomial with integer coefficients to fit the data. The coefficients are constrained to be positive.

Note that integer coefficients means that the resulting polynomial is of degree integer-1.

‘POLY = integer’

Uses a polynomial with integer coefficients to fit the data.
Note that integer coefficients means that the resulting polynomial is of degree \( \text{integer}-1 \).

'PADE= integer' Uses a Padé polynomial with integer coefficients to fit the data.

The Padé approximation used is not totally general, in that the same degree polynomial is used in the numerator and the denominator. The integer value supplied must be of the form \( 2n+1 \).

'OTW= integer ' For developer use only.

**Discussion and recommendations**

It has already been stated that ZIPPY2 is not capable of speeding up all runs and is most effective for those that chop repeatedly. ZIPPY2 can, however, be used when only a small part of the run is affected.

**Cosmetic Improvements**

As an alternative to using TUNING to reduce the maximum timestep before a problem starts and then untuning it afterwards, the ZIPPY2 keyword can be used to achieve a similar effect without the need to know what a workable maximum timestep is. For this situation the recommendation is simply to use Example 1 above.

**Examples**

**Example 1**

```plaintext
ZIPPY2
'SIM=3.1' 'MINSTEP=0.000001' /

ZIPPY2
'SIM=6.1' 'MINSTEP=0.000001' 'AGGRESSIVE=YES' /
```

It is strongly recommended that the ZIPP2OFF keyword is used once the problem period is over, otherwise there may be a significant loss of efficiency.

The very small value of MINSTEP entered is only to ensure that the simulator does not accept un-converged timesteps due to being unable to take smaller timesteps; the simulator will not take timesteps as small as this unless it really needs to. (In ECLIPSE 100, setting MINCHOP to a small value is also a good safety precaution.)

The value of 3.1 Newtons is a little conservative and, in most cases, will not be the most efficient. A more efficient value is probably between 4 and 5, but every case is different and it is impossible to generalize with any great success. Integer values should be avoided.

One strategy is to use ZIPPY2 to find what timestep values are appropriate. This information can then be used to guide the choice of where to place the TUNING keywords and what maximum timesteps can be reasonably set.

**Example 2**

It is strongly recommended that the ZIPP2OFF keyword is used once the problem period is over. There may otherwise be a significant loss of efficiency.

---

**ECLIPSE 100 only**

When the 'SIM' option is not being used the rest of the options attempt to represent the cost per simulated day as a function of the timestep taken. This function is then minimized to give the most efficient timestep.
To improve efficiency

ECLIPSE chops the timestep for one reason: because it failed to converge within the given number of Newton iterations. However, there are two reasons why this happens.

- The first case is when the Newton iteration may be diverging; in this case increasing the maximum number of Newtons means that more time is wasted.
- The second case is when the Newton iterations are converging, but do not make the set tolerances before the maximum Newton limit is reached.

It is sometimes possible to distinguish which might be the case by comparing the length of timestep with the number of Newtons needed. If the ratio of timesteps is roughly equal to the ratio of Newtons used then the case is probably the latter. If a small increase in timestep leads to an unexpectedly large growth in the number of Newtons then there may well be divergence.

Example 3 - Diverging Iterations

In this case it is best to start with the simple device of:

```
ZIPPY2
'SIM=3.1' /
```

However, this choice can be very inefficient for converging problems.

Example 4 – Converging Iterations or uncertainty

**ECLIPSE 100 only**

Whilst every case is different, one of the most robust choices for ECLIPSE 100 seems to be:

```
ZIPPY2
'CPU' 'NPTS=15' 'SPRING=3.1,2' 'BEZIER=5' /
```

If it is important to be able to repeat the time stepping then replace 'CPU' with something like 'SOLVER=4.1'.

The SPRING option is included to penalize high numbers of iterations. In a converging case we do not want to predict a timestep, no matter how efficient it is predicted to be, if it is going to take even one more Newton than is allowed; hence the inclusion of this artificial penalty function.

Clearly the number of possible combinations is endless. One could soon waste as much time with them as with trying to get the maximum timestep right with the TUNING keyword. The combinations most worthy of trying are:

- Adding 'NEWTON=20', for example.

  In the converging case this may allow larger timesteps to be taken if they are predicted to be more efficient. Beware, though, that when the code chops--none of these options guarantees no chopping--then with a higher maximum number of Newtons more time is wasted.

- Using 'BEZIER2=5'

  Sometimes, more usually in the diverging iteration case, 'BEZIER2=5' is better than 'BEZIER=5'.
**ZMF**

**Specifies cell initial total composition explicitly**

This keyword specifies the initial total composition values for each cell. The keyword should be followed by $N_x \cdot N_y \cdot N_z \cdot N_c$ values in normal order, with the X index changing fastest. For thermal runs with water, $N_x \cdot N_y \cdot N_z$ extra values are required specifying the initial water composition for each cell.

This keyword is normally used to specify the initial solution explicitly, as an alternative to equilibration using EQUIL, the usual method of initializing a compositional run. Explicit initialization will be expected if the PRESSURE keyword is present. The syntax follows that of XMF and YMF. In this context, the keyword should only be used when the reservoir contains a single hydrocarbon phase (either oil or gas).

An alternative use of the keyword is to overwrite the hydrocarbon composition in a case that has been equilibrated. This can be useful as the equilibration process generates the water saturations, but an engineer may want a non-equilibrium start to handle areal variations in composition. In this case the ZMF keyword is used with the EQUIL keyword (not with the PRESSURE keyword). If this option is used then the usual data for equilibration (for example ZMFDV) needs to be entered to obtain the initial equilibrium solution.

ZMF values that do not sum to 1.0 are normalized provided the error is less than 10%. An error greater than 10% is flagged as a data error.

**Example**

```plaintext
ZMF
324*0.0021
324*0.0294
324*0.6599
324*0.0869
324*0.0591
324*0.0967
324*0.0472
324*0.0153
324*0.0034 /
```

**Keywords**

2780
Total composition with respect to depth tables

The data comprises a table of total composition with respect to depth data for each equilibration region. Each table consists of \( N_c + 1 \) columns of data, and is terminated with a slash (/).

Column:

1. Depth values.

   The values should increase monotonically down the column.

   **UNITS:** m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

2. to \( N_c + 1 \) The corresponding values of \( z_i \), the total phase mole fractions. The \( N_c \) mole fractions specified must add up to unity, and the program will check that this is the case.

For depth values outside the interval specified, constant value extrapolation is performed. If the composition is independent of depth, only a single row is required.

The default maximum number of rows per table is 50; this may be reset using the third item in keyword **EQLDIMS**. The number of equilibration regions is entered using the first item in keyword **EQLDIMS**.

See also "Initial composition with respect to depth" in the **ECLIPSE Technical Description**.

Example

In this case a constant composition with respect to depth for 2 equilibration regions, with a nine-component system:

```
ZMFVD
1000.0  0.0121  0.0194  0.6597  0.0869  0.0591
   0.0967  0.0472  0.0153  0.0034
10000.0 0.0121  0.0194  0.6597  0.0869  0.0591
   0.0967  0.0472  0.0153  0.0034 /
1000.0  0.0121  0.0194  0.6597  0.0869  0.0591
   0.0967  0.0472  0.0153  0.0034
10000.0 0.0121  0.0194  0.6597  0.0869  0.0591
   0.0967  0.0472  0.0153  0.0034 /
```